build123d

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CONTENTS

Build123d is a Python-based, parametric (BREP) modeling framework for 2D and 3D CAD. Built on the Open Cascade geometric kernel, it provides a clean, fully Pythonic interface for creating precise models suitable for 3D printing, CNC machining, laser cutting, and other manufacturing processes. Models can be exported to popular CAD tools such as FreeCAD and SolidWorks.

Designed for modern, maintainable CAD-as-code, build123d combines clear architecture with expressive, algebraic modeling. It offers:

- Minimal or no internal state depending on mode
- Explicit 1D, 2D, and 3D geometry classes with well-defined operations
- Extensibility through subclassing and functional composition—no monkey patching
- Standards-compliant code (PEP 8, mypy, pylint) with rich pylance type hints
- Deep Python integration—selectors as lists, locations as iterables, and natural conversions (`Solid(shell)`, `tuple(Vector)`)
- Operator-driven modeling (`obj += sub_obj`, `Plane.XZ * Pos(X=5) * Rectangle(1, 1)`) for algebraic, readable, and composable design logic

With build123d, intricate parametric models can be created in just a few lines of readable Python code—as demonstrated by the tea cup example below.

Teacup Example

```python
from build123d import *
from ocp_vscode import show

wall_thickness = 3 * MM
fillet_radius = wall_thickness * 0.49

with BuildPart() as tea_cup:
    # Create the bowl of the cup as a revolved cross section
    with BuildSketch(Plane.XZ) as bowl_section:
        with BuildLine():
            # Start & end points with control tangents
            s = Spline(
                (30 * MM, 10 * MM),
                (69 * MM, 105 * MM),
                tangents=((1, 0.5), (0.7, 1)),
                tangent_scalars=(1.75, 1),
            )
            # Lines to finish creating ½ the bowl shape
            Polyline(s @ 0, s @ 0 + (10 * MM, -10 * MM), (0, 0), (0, (s @ 1).Y), s @ 1)
        make_face()  # Create a filled 2D shape
    revolve(axis=Axis.Z)
    # Hollow out the bowl with openings on the top and bottom
    offset(amount=-wall_thickness, openings=tea_cup.faces().filter_by(GeomType.PLANE))
    # Add a bottom to the bowl
    with Locations((0, 0, (s @ 0).Y)):
        Cylinder(radius=(s @ 0).X, height=wall_thickness)
    # Smooth out all the edges
    fillet(tea_cup.edges(), radius=fillet_radius)
```

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```
    # Determine where the handle contacts the bowl
    handle_intersections = [
        tea_cup.part.find_intersection_points(
            Axis(origin=(0, 0, vertical_offset), direction=(1, 0, 0))
        )[-1][0]
        for vertical_offset in [35 * MM, 80 * MM]
    ]
    # Create a path for handle creation
    with BuildLine(Plane.XZ) as handle_path:
        Spline(
            handle_intersections[0] - (wall_thickness / 2, 0),
            handle_intersections[0] + (35 * MM, 30 * MM),
            handle_intersections[0] + (40 * MM, 60 * MM),
            handle_intersections[1] - (wall_thickness / 2, 0),
            tangents=((1, 1.25), (-0.2, -1)),
        )
    # Align the cross section to the beginning of the path
    with BuildSketch(handle_path.line ^ 0) as handle_cross_section:
        RectangleRounded(wall_thickness, 8 * MM, fillet_radius)
    sweep()  # Sweep handle cross section along path

assert abs(tea_cup.part.volume - 130326) < 1

show(tea_cup, names=["tea cup"])
```

Note

This documentation is available in pdf and epub formats for reference while offline.

Note

There is a Discord server (shared with CadQuery) where you can ask for help in the build123d channel.

CHAPTER

ONE

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1.1 Introduction

1.1.1 Key Aspects

The following sections describe some of the key aspects of build123d and illustrate why one might choose this open source system over proprietary options like SolidWorks, OnShape, Fusion 360, or even other open source systems like Blender, or OpenSCAD.

Boundary Representation (BREP) Modelling

Boundary representation (BREP) and mesh-based CAD systems are both used to create and manipulate 3D models, but they differ in the way they represent and store the models.

Advantages of BREP-based CAD systems (e.g. build123d & SolidWorks):

- Precision: BREP-based CAD systems use mathematical representations to define the shape of an object, which allows for more precise and accurate modeling of complex shapes.
- Topology: BREP-based CAD systems maintain topological information of the 3D model, such as edges, faces and vertices. This allows for more robust and stable modeling, such as Boolean operations.
- Analytical modeling: BREP-based CAD systems can take advantage of the topological information to perform analytical operations such as collision detection, mass properties calculations, and finite element analysis.
- Features-based modeling: BREP-based CAD systems are often feature-based, which means that the model is built by creating and modifying individual features, such as holes, fillets, and chamfers. This allows for parametric design and easy modification of the model.
- Efficient storage: BREP-based CAD systems use a compact representation to store the 3D model, which is more efficient than storing a large number of triangles used in mesh-based systems.

Advantages of Mesh-based CAD systems (e.g. Blender, OpenSCAD):

- Simplicity: Mesh-based CAD systems use a large number of triangles to represent the surface of an object, which makes them easy to use and understand.
- Real-time rendering: Mesh-based CAD systems can be rendered in real-time, which is useful for applications such as video games and virtual reality.
- Flexibility: Mesh-based CAD systems can be easily exported to other 3D modeling and animation software, which makes them a good choice for use in the entertainment industry.
- Handling of freeform surfaces: Mesh-based systems are better equipped to handle freeform surfaces, such as those found in organic shapes, as they do not rely on mathematical representation.
- Handling of large datasets: Mesh-based systems are more suitable for handling large datasets such as point clouds, as they can be easily converted into a mesh representation.

Parameterized Models

Parametrized CAD systems are more effective than non-parametric CAD systems in several ways:

- Reusability: Parametrized CAD models can be easily modified by changing a set of parameters, such as the length or width of an object, rather than having to manually edit the geometry. This makes it easy to create variations of a design without having to start from scratch.
- Design exploration: Parametrized CAD systems allow for easy exploration of different design options by changing the parameters and quickly visualizing the results. This can save a lot of time and effort during the design process.
- Constraints and relationships: Parametrized CAD systems allow for the definition of constraints and relationships between different parameters. This ensures that the model remains valid and functional even when parameters are changed.
- Automation: Parametrized CAD systems can be automated to perform repetitive tasks, such as generating detailed drawings or creating parts lists. This can save a lot of time and effort and reduce the risk of errors.
- Collaboration: Parametrized CAD systems allow different team members to work on different aspects of a design simultaneously and ensure that the model remains consistent across different stages of the development process.
- Document management: Parametrized CAD systems can generate engineering drawings, BOMs, and other documents automatically, which makes it easier to manage and track the design history.

In summary, parametrized CAD systems are more effective than non-parametric CAD systems because they provide a more efficient and flexible way to create and modify designs, and can be easily integrated into the design, manufacturing, and documentation process.

Python Programming Language

Python is a popular, high-level programming language that has several advantages over other programming languages:

- Readability: Python code is easy to read and understand, with a clear and consistent syntax. This makes it a great language for beginners and for teams of developers who need to collaborate on a project.
- Versatility: Python is a general-purpose language that can be used for a wide range of tasks, including web development, scientific computing, data analysis, artificial intelligence, and more. This makes it a great choice for developers who need to work on multiple types of projects.
- Large community: Python has a large and active community of developers who contribute to the language and its ecosystem. This means that there are many libraries and frameworks available for developers to use, which can save a lot of time and effort.
- Good for data science, machine learning, and CAD: Python has a number of libraries such as numpy, pandas, scikit-learn, tensorflow, and cadquery which are popularly used in data science and machine learning and CAD.
- High-level language: Python is a high-level language, which means it abstracts away many of the low-level details of the computer. This makes it easy to write code quickly and focus on solving the problem at hand.
- Cross-platform: Python code runs on many different platforms, including Windows, Mac, and Linux, making it a great choice for developers who need to write code that runs on multiple operating systems.
- Open-source: Python is an open-source programming language, which means it is free to use and distribute. This makes it accessible to developers of all levels and budgets.
- Large number of libraries and modules: Python has a vast collection of libraries and modules that make it easy to accomplish complex tasks such as connecting to web servers, reading and modifying files, and connecting to databases.

Open Source Software

Open source and proprietary software systems are different in several ways: B Licensing: Open source software is licensed in a way that allows users to view, modify, and distribute the source code, while proprietary software is closed source and the source code is not available to the public.

- Ownership: Open source software is usually developed and maintained by a community of developers, while proprietary software is owned by a company or individual.
- Cost: Open source software is typically free to use, while proprietary software may require payment for a license or subscription. Customization: Open source software can be modified and customized by users and developers, while proprietary software is typically not modifiable by users.
- Support: Open source software may have a larger community of users who can provide support, while proprietary software may have a smaller community and relies on the company for support. Security: Open source software can be audited by a large community of developers, which can make it more secure, while proprietary software may have fewer eyes on the code and may be more vulnerable to security issues.
- Interoperability: Open source software may have better interoperability with other software and platforms, while proprietary software may have more limited compatibility.
- Reliability: Open source software can be considered as reliable as proprietary software. It is usually used by large companies, governments, and organizations and has been tested by a large number of users.

In summary, open source and proprietary software systems are different in terms of licensing, ownership, cost, customization, support, security, interoperability, and reliability. Open source software is typically free to use and can be modified by users and developers, while proprietary software is closed-source and may require payment for a license or subscription. Open source software may have a larger community of users who can provide support, while proprietary software may have a smaller community and relies on the company for support.

Source Code Control Systems

Most GUI based CAD systems provide version control systems which represent the CAD design and its history. They allows developers to see changes made to the design over time, in a format that is easy to understand.

On the other hand, a source code control system like Git, is a command-line tool and it provides more granular control over the code. This makes it suitable for more advanced users and developers who are comfortable working with command-line interfaces. A source code control system like Git is more flexible and allows developers to perform tasks like branching and merging, which are not easily done with a GUI version control system. Systems like Git have several advantages, including:

- Version control: Git allows developers to keep track of changes made to the code over time, making it easy to revert to a previous version if necessary.
- Collaboration: Git makes it easy for multiple developers to work on the same codebase simultaneously, with the ability to merge changes from different branches of development.
- Backup: Git provides a way to backup and store the codebase in a remote repository, like GitHub. This can serve as a disaster recovery mechanism, in case of data loss.
- Branching: Git allows developers to create multiple branches of a project for different features or bug fixes, which can be easily merged into the main codebase once they are complete.
- Auditing: Git allows you to see who made changes to the code, when and what changes were made, which is useful for auditing and debugging.
- Open-source development: Git makes it easy for open-source developers to contribute to a project and share their work with the community.
- Flexibility: Git is a distributed version control system, which means that developers can work independently and offline. They can then push their changes to a remote repository when they are ready to share them with others.

In summary, GUI version control systems are generally more user-friendly and easier to use, while source code control systems like Git offer more flexibility and control over the code. Both can be used to achieve the same goal, but they cater to different types of users and use cases.

Automated Testing

Users of source based CAD systems can benefit from automated testing which improves their source code by:

- Finding bugs: Automated tests can detect bugs in the code, which can then be fixed before the code is released. This helps to ensure that the code is of higher quality and less likely to cause issues when used.
- Regression testing: Automated tests can be used to detect regressions, which are bugs that are introduced by changes to the codebase. This helps to ensure that changes to the code do not break existing functionality.
- Documenting code behavior: Automated tests can serve as documentation for how the code is supposed to behave. This makes it easier for developers to understand the code and make changes without breaking it.
- Improving code design: Writing automated tests often requires a good understanding of the code and how it is supposed to behave. This can lead to a better design of the code, as developers will have a better understanding of the requirements and constraints.
- Saving time and cost: Automated testing can save time and cost by reducing the need for manual testing. Automated tests can be run quickly and often, which means that bugs can be found and fixed early in the development process, which is less expensive than finding them later.
- Continuous integration and delivery: Automated testing can be integrated into a continuous integration and delivery (CI/CD) pipeline. This means that tests are run automatically every time code is committed and can be integrated with other tools such as code coverage, static analysis and more.
- Improving maintainability: Automated tests can improve the maintainability of the code by making it easier to refactor and change the codebase. This is because automated tests provide a safety net that ensures that changes to the code do not introduce new bugs.

Overall, automated testing is an essential part of the software development process, it helps to improve the quality of the code by detecting bugs early, documenting code behavior, and reducing the cost of maintaining and updating the code.

Automated Documentation

The Sphinx automated documentation system was used to create the page you are reading now and can be used for user design documentation as well. Such systems are used for several reasons:

- Consistency: Sphinx and other automated documentation systems can generate documentation in a consistent format and style, which makes it easier to understand and use.
- Automation: Sphinx can automatically generate documentation from source code and comments, which saves time and effort compared to manually writing documentation.
- Up-to-date documentation: Automated documentation systems like Sphinx can update the documentation automatically when the code changes, ensuring that the documentation stays up-to-date with the code.
- Searchability: Sphinx and other automated documentation systems can include search functionality, which makes it easy to find the information you need.
- Cross-referencing: Sphinx can automatically create links between different parts of the documentation, making it easy to navigate and understand the relationships between different parts of the code.
- Customizable: Sphinx and other automated documentation systems can be customized to match the look and feel of your company's documentation.
- Multiple output formats: Sphinx can generate documentation in multiple formats such as HTML, PDF, ePub, and more.

- Support for multiple languages: Sphinx can generate documentation in multiple languages, which can make it easier to support international users.
- Integration with code management: Sphinx can be integrated with code management tools like Git, which allows documentation to be versioned along with the code.

In summary, automated documentation systems like Sphinx are used to generate consistent, up-to-date, and searchable documentation from source code and comments. They save time and effort compared to manual documentation, and can be customized to match the look and feel of your company's documentation. They also provide multiple output formats, support for multiple languages and can be integrated with code management tools.

1.1.2 Advantages Over CadQuery

As mentioned previously, the most significant advantage is that build123d is more pythonic. Specifically:

Standard Python Context Manager

The creation of standard instance variables, looping and other normal python operations is enabled by the replacement of method chaining (fluent programming) with a standard python context manager.

```
# CadQuery Fluent API
pillow_block = (cq.Workplane("XY")
    .box(height, width, thickness)
    .edges("|Z")
    .fillet(fillet)
    .faces(">Z")
    .workplane()
    ...
)
```

```
# build123d API
with BuildPart() as pillow_block:
    with BuildSketch() as plan:
        Rectangle(width, height)
        fillet(plan.vertices(), radius=fillet)
    extrude(thickness)
    ...
```

The use of the standard *with* block allows standard python instructions to be inserted anyway in the code flow. One can insert a CQ-editor *debug* or standard *print* statement anywhere in the code without impacting functionality. Simple python *for* loops can be used to repetitively create objects instead of forcing users into using more complex *lambda* and *iter* operations.

Instantiated Objects

Each object and operation is now a class instantiation that interacts with the active context implicitly for the user. These instantiations can be assigned to an instance variable as with standard python programming for direct use.

```
with BuildSketch() as plan:
    r = Rectangle(width, height)
    print(r.area)
    ...
```

Operators

New operators have been created to extract information from objects created previously in the code. The @ operator extracts the position along an Edge or Wire while the % operator extracts the tangent along an Edge or Wire. The position parameter are float values between 0.0 and 1.0 which represent the beginning and end of the line. In the following example, a spline is created from the end of l5 (*l5 @ 1*) to the beginning of l6 (*l6 @ 0*) with tangents equal to the tangents of l5 and l6 at their end and beginning respectively. Being able to extract information from existing features allows the user to "snap" new features to these points without knowing their numeric values.

```
with BuildLine() as outline:
    ...
    l5 = Polyline(...)
    l6 = Polyline(...)
    Spline(l5 @ 1, l6 @ 0, tangents=(l5 % 1, l6 % 0))
```

Last Operation Objects

All of the *vertices()*, *edges()*, *faces()*, and *solids()* methods of the builders can either return all of the objects requested or just the objects changed during the last operation. This allows the user to easily access features for further refinement, as shown in the following code where the final line selects the edges that were added by the last operation and fillets them. Such a selection would be quite difficult otherwise.

```
from build123d import *

with BuildPart() as pipes:
    box = Box(10, 10, 10, rotation=(10, 20, 30))
    with BuildSketch(*box.faces()) as pipe:
        Circle(4)
    extrude(amount=-5, mode=Mode.SUBTRACT)
    with BuildSketch(*box.faces()) as pipe:
        Circle(4.5)
        Circle(4, mode=Mode.SUBTRACT)
    extrude(amount=10)
    fillet(pipes.edges(Select.LAST), 0.2)
```

Extensions

Extending build123d is relatively simple in that custom objects or operations can be created as new classes without the need to monkey patch any of the core functionality. These new classes will be seen in IDEs which is not possible with monkey patching the core CadQuery classes.

Enums

All *Literal* strings have been replaced with *Enum* which allows IDEs to prompt users for valid options without having to refer to documentation.

Selectors replaced by Lists

String based selectors have been replaced with standard python filters and sorting which opens up the full functionality of python lists. To aid the user, common operations have been optimized as shown here along with a fully custom selection:

```
top = rail.faces().filter_by(Axis.Z)[-1]
...
```

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```
outside_vertices = filter(
    lambda v: (v.Y == 0.0 or v.Y == height) and -width / 2 < v.X < width / 2,
    din.vertices(),
)
```

1.2 Installation

The recommended method for most users to install **build123d** is:

```
>>> pip install build123d
```

Note

The ocp-vscode viewer has the ability to install **build123d**.

1.2.1 Install build123d from GitHub:

To get the latest non-released version of **build123d** one can install from GitHub using one of the following two commands:

In Linux/MacOS, use the following command:

```
>>> python3 -m pip install git+https://github.com/gumyr/build123d
```

In Windows, use the following command:

```
>>> python -m pip install git+https://github.com/gumyr/build123d
```

If you receive errors about conflicting dependencies, you can retry the installation after having upgraded pip to the latest version with the following command:

```
>>> python3 -m pip install --upgrade pip
```

If you use poetry to install build123d, you can simply use:

```
>>> poetry add build123d
```

However, if you want the latest commit from GitHub you might need to specify the branch that is used for git-based installs; until quite recently, poetry used to checkout the *master* branch when none was specified, and this fails on build123d that uses a *dev* branch.

Pip does not suffer from this issue because it correctly fetches the repository default branch.

If you are a poetry user, you can work around this issue by installing build123d in the following way:

```
>>> poetry add git+https://github.com/gumyr/build123d.git@dev
```

Please note that always suffixing the URL with `@dev` is safe and will work with both older and recent versions of poetry.

1.2.2 Development install of build123d:

Warning: it is highly recommended to upgrade pip to the latest version before installing build123d, especially in development mode. This can be done with the following command:

```
>>> python3 -m pip install --upgrade pip
```

Once pip is up-to-date, you can install build123d in development mode with the following commands:

```
>>> git clone https://github.com/gumyr/build123d.git
>>> cd build123d
>>> python3 -m pip install -e .
```

Please substitute `python3` with `python` in the lines above if you are using Windows.

If you're working directly with the OpenCascade `OCP` layer you will likely want to install the OCP stubs as follows:

```
>>> python3 -m pip install git+https://github.com/CadQuery/OCP-stubs@7.7.0
```

1.2.3 Test your build123d installation:

If all has gone well, you can open a command line/prompt, and type:

```
>>> python
>>> from build123d import *
>>> print(Solid.make_box(1,2,3).show_topology(limit_class="Face"))
```

Which should return something similar to:

```
Solid        at 0x165e75379f0, Center(0.5, 1.0, 1.5)
└── Shell    at 0x165eab056f0, Center(0.5, 1.0, 1.5)
        ├── Face at 0x165b35a3570, Center(0.0, 1.0, 1.5)
        ├── Face at 0x165e77957f0, Center(1.0, 1.0, 1.5)
        ├── Face at 0x165b3e730f0, Center(0.5, 0.0, 1.5)
        ├── Face at 0x165e8821570, Center(0.5, 2.0, 1.5)
        ├── Face at 0x165e88218f0, Center(0.5, 1.0, 0.0)
        └── Face at 0x165eb21ee70, Center(0.5, 1.0, 3.0)
```

1.2.4 Adding a nicer GUI

If you prefer to have a GUI available, your best option is to choose one from here: *External Tools and Libraries*

1.3 Key Concepts

The following key concepts will help new users understand build123d quickly.

1.3.1 Topology

Topology, in the context of 3D modeling and computational geometry, is the branch of mathematics that deals with the properties and relationships of geometric objects that are preserved under continuous deformations. In the context of CAD and modeling software like build123d, topology refers to the hierarchical structure of geometric elements (vertices, edges, faces, etc.) and their relationships in a 3D model. This structure defines how the components of a model are connected, enabling operations like Boolean operations, transformations, and analysis of complex shapes. Topology provides a formal framework for understanding and manipulating geometric data in a consistent and reliable manner.

The following are the topological objects that compose build123d objects:

Vertex
: A Vertex is a data structure representing a 0D topological element. It defines a precise point in 3D space, often at the endpoints or intersections of edges in a 3D model. These vertices are part of the topological structure used to represent complex shapes in build123d.

Edge
: An Edge in build123d is a fundamental geometric entity representing a 1D element in a 3D model. It defines the shape and position of a 1D curve within the model. Edges play a crucial role in defining the boundaries of faces and in constructing complex 3D shapes.

Wire
: A Wire in build123d is a topological construct that represents a connected sequence of Edges, forming a 1D closed or open loop within a 3D model. Wires define the boundaries of faces and can be used to create complex shapes, making them essential for modeling in build123d.

Face
: A Face in build123d represents a 2D surface in a 3D model. It defines the boundary of a region and can have associated geometric and topological data. Faces are vital for shaping solids, providing surfaces where other elements like edges and wires are connected to form complex structures.

Shell
: A Shell in build123d represents a collection of Faces, defining a closed, connected volume in 3D space. It acts as a container for organizing and grouping faces into a single shell, essential for defining complex 3D shapes like solids or assemblies within the build123d modeling framework.

Solid
: A Solid in build123d is a 3D geometric entity that represents a bounded volume with well-defined interior and exterior surfaces. It encapsulates a closed and watertight shape, making it suitable for modeling solid objects and enabling various Boolean operations such as union, intersection, and subtraction.

Compound
: A Compound in build123d is a container for grouping multiple geometric shapes. It can hold various types of entities, such as vertices, edges, wires, faces, shells, or solids, into a single structure. This makes it a versatile tool for managing and organizing complex assemblies or collections of shapes within a single container.

Shape
: A Shape in build123d represents a fundamental building block in 3D modeling. It encompasses various topological elements like vertices, edges, wires, faces, shells, solids, and compounds. The Shape class is the base class for all of the above topological classes.

One can use the *show_topology()* method to display the topology of a shape as shown here for a unit cube:

```
Solid                    at 0x7f94c55430f0, Center(0.5, 0.5, 0.5)
└── Shell                at 0x7f94b95835f0, Center(0.5, 0.5, 0.5)
    ├── Face             at 0x7f94b95836b0, Center(0.0, 0.5, 0.5)
    │   └── Wire         at 0x7f94b9583730, Center(0.0, 0.5, 0.5)
    │       ├── Edge     at 0x7f94b95838b0, Center(0.0, 0.0, 0.5)
    │       │   ├── Vertex at 0x7f94b9583470, Center(0.0, 0.0, 1.0)
    │       │   └── Vertex at 0x7f94b9583bb0, Center(0.0, 0.0, 0.0)
    │       ├── Edge     at 0x7f94b9583a30, Center(0.0, 0.5, 1.0)
    │       │   ├── Vertex at 0x7f94b9583030, Center(0.0, 1.0, 1.0)
    │       │   └── Vertex at 0x7f94b9583e70, Center(0.0, 0.0, 1.0)
    │       ├── Edge     at 0x7f94b9583770, Center(0.0, 1.0, 0.5)
    │       │   ├── Vertex at 0x7f94b9583bb0, Center(0.0, 1.0, 1.0)
    │       │   └── Vertex at 0x7f94b9583e70, Center(0.0, 1.0, 0.0)
    │       └── Edge     at 0x7f94b9583db0, Center(0.0, 0.5, 0.0)
```

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```
    │           ├── Vertex at 0x7f94b9583e70, Center(0.0, 1.0, 0.0)
    │           └── Vertex at 0x7f94b95862f0, Center(0.0, 0.0, 0.0)
...
    └── Face              at 0x7f94b958d3b0, Center(0.5, 0.5, 1.0)
        └── Wire          at 0x7f94b958d670, Center(0.5, 0.5, 1.0)
            ├── Edge      at 0x7f94b958e130, Center(0.0, 0.5, 1.0)
            │   ├── Vertex at 0x7f94b958e330, Center(0.0, 1.0, 1.0)
            │   └── Vertex at 0x7f94b958e770, Center(0.0, 0.0, 1.0)
            ├── Edge      at 0x7f94b958e630, Center(0.5, 1.0, 1.0)
            │   ├── Vertex at 0x7f94b958e8b0, Center(1.0, 1.0, 1.0)
            │   └── Vertex at 0x7f94b958ea70, Center(0.0, 1.0, 1.0)
            ├── Edge      at 0x7f94b958e7b0, Center(1.0, 0.5, 1.0)
            │   ├── Vertex at 0x7f94b958ebb0, Center(1.0, 1.0, 1.0)
            │   └── Vertex at 0x7f94b958ed70, Center(1.0, 0.0, 1.0)
            └── Edge      at 0x7f94b958eab0, Center(0.5, 0.0, 1.0)
                ├── Vertex at 0x7f94b958eeb0, Center(1.0, 0.0, 1.0)
                └── Vertex at 0x7f94b9592170, Center(0.0, 0.0, 1.0)
```

Users of build123d will often reference topological objects as part of the process of creating the object as described below.

1.3.2 Location

A *Location* represents a combination of translation and rotation applied to a topological or geometric object. It encapsulates information about the spatial orientation and position of a shape within its reference coordinate system. This allows for efficient manipulation of shapes within complex assemblies or transformations. The location is typically used to position shapes accurately within a 3D scene, enabling operations like assembly, and boolean operations. It's an essential component in build123d for managing the spatial relationships of geometric entities, providing a foundation for precise 3D modeling and engineering applications.

The topological classes (sub-classes of *Shape*) and the geometric classes *Axis* and *Plane* all have a `location` property. The *Location* class itself has `position` and `orientation` properties that have setters and getters as shown below:

```
>>> from build123d import *
>>> # Create an object and extract its location
>>> b = Box(1, 1, 1)
>>> box_location = b.location
>>> box_location
(p=(0.00, 0.00, 0.00), o=(-0.00, 0.00, -0.00))
>>> # Set position and orientation independently
>>> box_location.position = (1, 2, 3)
>>> box_location.orientation = (30, 40, 50)
>>> box_location.position
Vector: (1.0, 2.0, 3.0)
>>> box_location.orientation
Vector: (29.999999999999993, 40.00000000000002, 50.000000000000036)
```

Combining the getter and setter enables relative changes as follows:

```
>>> # Relative change
>>> box_location.position += (3, 2, 1)
>>> box_location.position
```

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```
Vector: (4.0, 4.0, 4.0)
```

There are also four methods that are used to change the location of objects:

- *locate()* - absolute change of this object
- *located()* - absolute change of copy of this object
- *move()* - relative change of this object
- *moved()* - relative change of copy of this object

Locations can be combined with the * operator and have their direction flipped with the - operator.

1.3.3 Selectors

When using a GUI based CAD system the user will often click on a feature to select it for some operation. How does a user "click" when CAD is done entirely in code? Selectors are recipes for how to isolate a feature from a design using python filter and sorting methods typically implemented as a set of custom python operations.

Quick Reference

The following tables describes the build123d selectors:

Selector	Applicability	Description	Example
vertices()	BuildLine, BuildSketch, BuildPart	Vertex extraction	*part.vertices()*
edges()	BuildLine, BuildSketch, BuildPart	Edge extraction	*part.edges()*
wires()	BuildLine, BuildSketch, BuildPart	Wire extraction	*part.wires()*
faces()	BuildSketch, BuildPart	Face extraction	*part.faces()*
solids()	BuildPart	Solid extraction	*part.solids()*

Operator	Operand	Method	Description	Example
>	SortBy, Axis	sort_by	Sort ShapeList by operand	*part.vertices() > Axis.Z*
<	SortBy, Axis	sort_by	Reverse sort ShapeList by operand	*part.faces() < Axis.Z*
>>	SortBy, Axis	group_by	Group ShapeList by operand and return last value	*part.solids() >> Axis.X*
<<	SortBy, Axis	group_by	Group ShapeList by operand and return first value	*part.faces() << Axis.Y*
|	Axis, Plane, GeomType	filter_by	Filter and sort ShapeList by Axis, Plane, or GeomType	*part.faces() | Axis.Z*
[]			Standard python list indexing and slicing	*part.faces()[-2:]*
	Axis	filter_by_posi	Filter ShapeList by Axis & mix / max values	*part.faces()..filter_by_position(Axis.Z, 1, 2, inclusive=(False, True))*

The operand types are: Axis, Plane, SortBy, and GeomType. An Axis is a base object with an origin and a direction with several predefined values such as `Axis.X`, `Axis.Y`, and `Axis.Z`; however, any Axis could be used as an operand (e.g. `Axis((1,2,3),(0.5,0,-0.5))` is valid) - see *Axis* for a complete description. A Plane is a coordinate system defined by an origin, x_dir (X direction), y_dir (Y direction), and z_dir (Z direction). See *Plane* for a complete

description. Filtering by a Plane will return faces/edges parallel to it. SortBy and GeomType are python Enum class described here:

GeomType
: BEZIER, BSPLINE, CIRCLE, CONE, CYLINDER, ELLIPSE, EXTRUSION, HYPERBOLA, LINE, OFFSET, OTHER, PARABOLA, PLANE, REVOLUTION, SPHERE, TORUS

SortBy
: LENGTH, RADIUS, AREA, VOLUME, DISTANCE

ShapeList Class

The builders include methods to extract Edges, Faces, Solids, Vertices, or Wires from the objects they are building. All of these methods return objects of a subclass of *list*, a *ShapeList* with custom filtering and sorting methods and operations as follows.

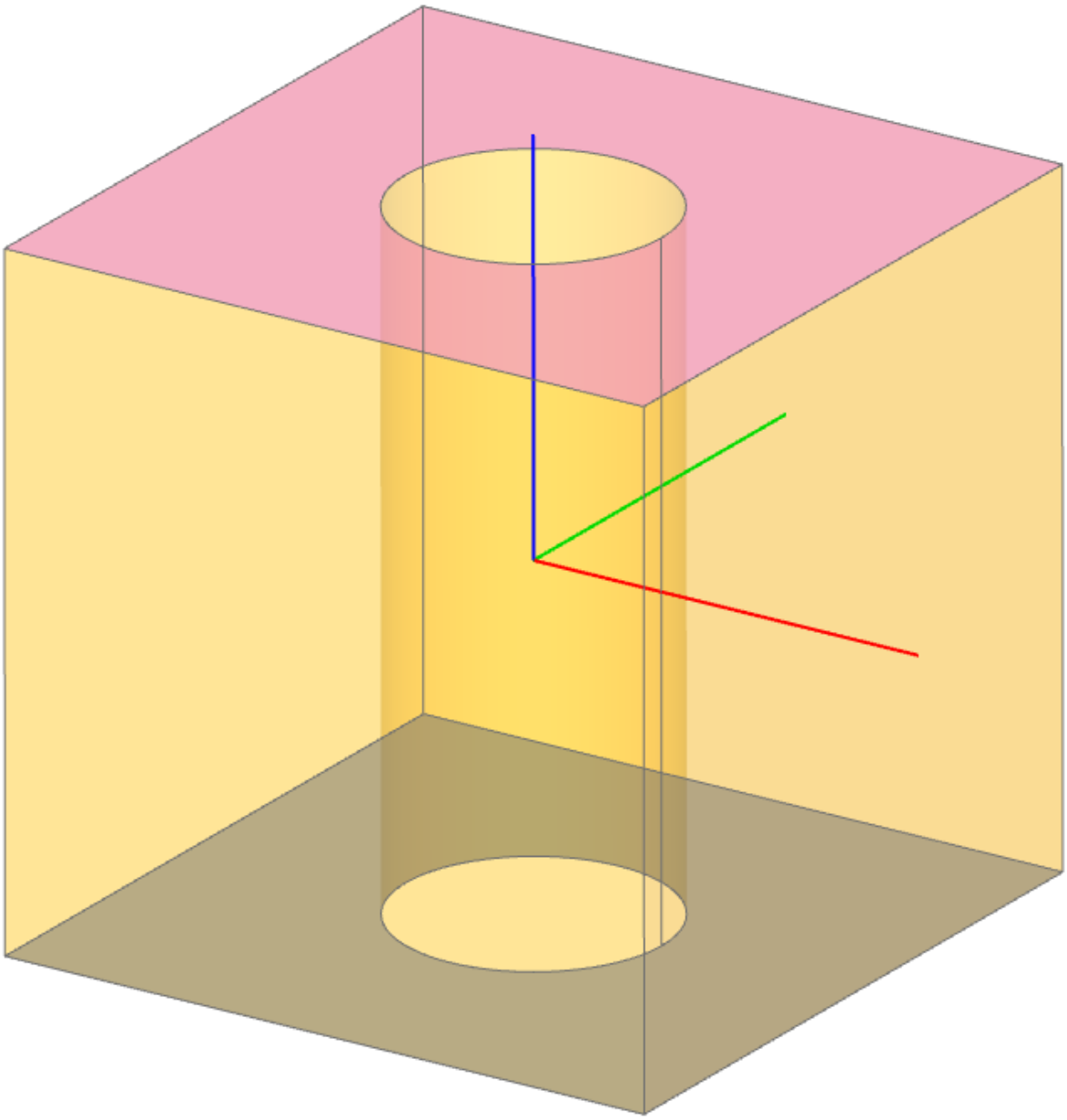
Custom Sorting and Filtering

It is important to note that standard list methods such as *sorted* or *filtered* can be used to easily build complex selectors beyond what is available with the predefined sorts and filters. Here is an example of a custom filters:

```
with BuildSketch() as din:
    ...
    outside_vertices = filter(
        lambda v: (v.Y == 0.0 or v.Y == height)
        and -overall_width / 2 < v.X < overall_width / 2,
        din.vertices(),
    )
```

The *filter_by()* method can take lambda expressions as part of a fluent chain of operations which enables integration of custom filters into a larger change of selectors as shown in this example:

```
obj = Box(1, 1, 1) - Cylinder(0.2, 1)
faces_with_holes = obj.faces().filter_by(lambda f: f.inner_wires())
```



Here the two faces with “inner_wires” (i.e. holes) have been selected independent of orientation.

1.4 Key Concepts (builder mode)

There are two primary APIs provided by build123d: builder and algebra. The builder API may be easier for new users as it provides some assistance and shortcuts; however, if you know what a Quaternion is you might prefer the algebra API which allows CAD objects to be created in the style of mathematical equations. Both API can be mixed in the same model with the exception that the algebra API can’t be used from within a builder context. As with music, there is no “best” genre or API, use the one you prefer or both if you like.

The following key concepts will help new users understand build123d quickly.

1.4.1 Understanding the Builder Paradigm

The **Builder** paradigm in build123d provides a powerful and intuitive way to construct complex geometric models. At its core, the Builder works like adding a column of numbers on a piece of paper: a running "total" is maintained internally as each new object is added or modified. This approach simplifies the process of constructing models by breaking it into smaller, incremental steps.

How the Builder Works

When using a Builder (such as **BuildLine**, **BuildSketch**, or **BuildPart**), the following principles apply:

1. **Running Total**: - The Builder maintains an internal "total," which represents the current state of the object being built. - Each operation updates this total by combining the new object with the existing one.
2. **Combination Modes**: - Just as numbers in a column may have a + or - sign to indicate addition or subtraction, Builders use **modes** to control how each object is combined with the current total. - Common modes include:
   - **ADD**: Adds the new object to the current total.
   - **SUBTRACT**: Removes the new object from the current total.
   - **INTERSECT**: Keeps only the overlapping regions of the new object and the current total.
   - **REPLACE**: Entirely replace the running total.
   - **PRIVATE**: Don't change the running total at all.
   - The mode can be set dynamically for each operation, allowing for flexible and precise modeling.
3. **Extracting the Result**: - At the end of the building process, the final object is accessed through the Builder's attributes, such as `.line`, `.sketch`, or `.part`, depending on the Builder type. - For example:
   - **BuildLine**: Use `.line` to retrieve the final wireframe geometry.
   - **BuildSketch**: Use `.sketch` to extract the completed 2D profile.
   - **BuildPart**: Use `.part` to obtain the 3D solid.

Example Workflow

Here is an example of using a Builder to create a simple part:

```
from build123d import *

# Using BuildPart to create a 3D model
with BuildPart() as example_part:
    with BuildSketch() as base_sketch:
        Rectangle(20, 20)
    extrude(amount=10)  # Create a base block
    with BuildSketch(Plane(example_part.faces().sort_by(Axis.Z).last)) as cut_sketch:
        Circle(5)
    extrude(amount=-5, mode=Mode.SUBTRACT)  # Subtract a cylinder

# Access the final part
result_part = example_part.part
```

Key Concepts

- **Incremental Construction**: Builders allow you to build objects step-by-step, maintaining clarity and modularity.
- **Dynamic Mode Switching**: The **mode** parameter gives you precise control over how each operation modifies the current total.
- **Seamless Extraction**: The Builder paradigm simplifies the retrieval of the final object, ensuring that you always have access to the most up-to-date result.

Analogy: Adding Numbers on Paper

Think of the Builder as a running tally when adding numbers on a piece of paper:

- Each number represents an operation or object.
- The + or - sign corresponds to the **ADD** or **SUBTRACT** mode.
- At the end, the total is the sum of all operations, which you can retrieve by referencing the Builder's output.

By adopting this approach, build123d ensures a natural, intuitive workflow for constructing 2D and 3D models.

Note

Why modifying objects directly doesn't work in Builder mode

A common mistake in Builder mode is attempting to modify an object after it is created:

```
with BuildPart() as invalid:
    Cylinder(1, 2).moved(Location((1, 2, 3)))
```

Builder mode works by having objects add themselves to the active builder immediately when they are created. In the example above:

`Cylinder(1, 2)` creates the cylinder.

The cylinder immediately adds itself to the `BuildPart` builder.

`.moved(...)` is then applied to the temporary Python object returned by `Cylinder`.

Because the cylinder was already added to the builder, the move operation has no effect on the model being built.

Placement must therefore be specified before the object is created, which is why Builder mode provides the `Locations` context (see below):

```
with BuildPart() as valid:
    with Locations((1, 2, 3)):
        Cylinder(1, 2)
```

Here the builder knows the location before the cylinder is created, so the part is placed correctly.

A similar situation in normal Python

```
with open("test.txt", "w") as f:
    f.write("text").to_bytes(1, "big")
```

`f.write("text")` writes "text" to the file and returns 4. `.to_bytes(1, "big")` is then called on that integer, producing b"x04".

The file still contains only "text" because the additional operation happens after the write and its result is discarded.

Builder mode behaves similarly: once an object has been added to the builder, modifying the returned Python object does not change what was already added.

1.4.2 Builders

The three builders, `BuildLine`, `BuildSketch`, and `BuildPart` are tools to create new objects - not the objects themselves. Each of the objects and operations applicable to these builders create objects of the standard CadQuery Direct API, most commonly `Compound` objects. This is opposed to CadQuery's Fluent API which creates objects of the `Workplane` class which frequently needed to be converted back to base class for further processing.

One can access the objects created by these builders by referencing the appropriate instance variable. For example:

```
with BuildPart() as my_part:
    ...

show_object(my_part.part)
```

```
with BuildSketch() as my_sketch:
    ...

show_object(my_sketch.sketch)
```

```
with BuildLine() as my_line:
    ...

show_object(my_line.line)
```

1.4.3 Implicit Builder Instance Variables

One might expect to have to reference a builder's instance variable when using objects or operations that impact that builder like this:

```
with BuildPart() as part_builder:
    Box(part_builder, 10,10,10)
```

Instead, build123d determines from the scope of the object or operation which builder it applies to thus eliminating the need for the user to provide this information - as follows:

```
with BuildPart() as part_builder:
    Box(10,10,10)
    with BuildSketch() as sketch_builder:
        Circle(2)
```

In this example, `Box` is in the scope of `part_builder` while `Circle` is in the scope of `sketch_builder`.

1.4.4 Workplanes

As build123d is a 3D CAD package one must be able to position objects anywhere. As one frequently will work in the same plane for a sequence of operations, the first parameter(s) of the builders is a (sequence of) workplane(s) which is (are) used to aid in the location of features. The default workplane in most cases is the `Plane.XY` where a tuple of numbers represent positions on the x and y axes. However workplanes can be generated on any plane which allows users to put a workplane where they are working and then work in local 2D coordinate space.

```
with BuildPart(Plane.XY) as example:
    ... # a 3D-part
    with BuildSketch(example.faces().sort_by(sort_by=Axis.Z)[0]) as bottom:
        ...
```

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```
    with BuildSketch(Plane.XZ) as vertical:
        ...
    with BuildSketch(example.faces().sort_by(sort_by=Axis.Z)[-1]) as top:
        ...
```

When `BuildPart` is invoked it creates the workplane provided as a parameter (which has a default of the `Plane.XY`). The `bottom` sketch is therefore created on the `Plane.XY` but with the normal reversed to point down. Subsequently the user has created the `vertical` (`Plane.XZ`) sketch. All objects or operations within the scope of a workplane will automatically be orientated with respect to this plane so the user only has to work with local coordinates.

As shown above, workplanes can be created from faces as well. The `top` sketch is positioned on top of `example` by selecting its faces and finding the one with the greatest z value.

One is not limited to a single workplane at a time. In the following example all six faces of the first box are used to define workplanes which are then used to position rotated boxes.

```
import build123d as bd

with bd.BuildPart() as bp:
    bd.Box(3, 3, 3)
    with bd.BuildSketch(*bp.faces()):
        bd.Rectangle(1, 2, rotation=45)
    bd.extrude(amount=0.1)
```

This is the result:

1.4.5 Locations Context

When positioning objects or operations within a builder, Location Contexts are used. These function in a very similar way to the builders in that they create a context where one or more locations are active within a scope. For example:

```
with BuildPart():
    with Locations((0,10),(0,-10)):
        Box(1,1,1)
        with GridLocations(x_spacing=5, y_spacing=5, x_count=2, y_count=2):
            Sphere(1)
        Cylinder(1,1)
```

In this example `Locations` creates two positions on the current workplane at (0,10) and (0,-10). Since `Box` is within the scope of `Locations`, two boxes are created at these locations. The `GridLocations` context creates four positions which apply to the `Sphere`. The `Cylinder` is out of the scope of `GridLocations` but in the scope of `Locations` so two cylinders are created.

Note that these contexts are creating Location objects not just simple points. The difference isn't obvious until the `PolarLocations` context is used which can also rotate objects within its scope - much as the hour and minute indicator on an analogue clock.

Also note that the locations are local to the current location(s) - i.e. `Locations` can be nested. It's easy for a user to retrieve the global locations:

```
with Locations(Plane.XY, Plane.XZ):
    locs = GridLocations(1, 1, 2, 2)
    for l in locs:
        print(l)
```

```
Location(p=(-0.50,-0.50,0.00), o=(0.00,-0.00,0.00))
Location(p=(-0.50,0.50,0.00), o=(0.00,-0.00,0.00))
Location(p=(0.50,-0.50,0.00), o=(0.00,-0.00,0.00))
Location(p=(0.50,0.50,0.00), o=(0.00,-0.00,0.00))
Location(p=(-0.50,-0.00,-0.50), o=(90.00,-0.00,0.00))
Location(p=(-0.50,0.00,0.50), o=(90.00,-0.00,0.00))
Location(p=(0.50,0.00,-0.50), o=(90.00,-0.00,0.00))
Location(p=(0.50,0.00,0.50), o=(90.00,-0.00,0.00))
```

1.4.6 Operation Inputs

When one is operating on an existing object, e.g. adding a fillet to a part, an iterable of objects is often required (often a ShapeList).

Here is the definition of *fillet()* to help illustrate:

```
def fillet(
    objects: Union[Union[Edge, Vertex], Iterable[Union[Edge, Vertex]]],
    radius: float,
):
```

To use this fillet operation, an edge or vertex or iterable of edges or vertices must be provided followed by a fillet radius with or without the keyword as follows:

```
with BuildPart() as pipes:
    Box(10, 10, 10, rotation=(10, 20, 30))
    ...
    fillet(pipes.edges(Select.LAST), radius=0.2)
```

Here the fillet accepts the iterable ShapeList of edges from the last operation of the `pipes` builder and a radius is provided as a keyword argument.

1.4.7 Combination Modes

Almost all objects or operations have a `mode` parameter which is defined by the `Mode` Enum class as follows:

```
class Mode(Enum):
    ADD = auto()
    SUBTRACT = auto()
    INTERSECT = auto()
    REPLACE = auto()
    PRIVATE = auto()
```

The `mode` parameter describes how the user would like the object or operation to interact with the object within the builder. For example, `Mode.ADD` will integrate a new object(s) in with an existing `part`. Note that a part doesn't necessarily have to be a single object so multiple distinct objects could be added resulting is multiple objects stored as a `Compound` object. As one might expect `Mode.SUBTRACT`, `Mode.INTERSECT`, and `Mode.REPLACE` subtract, intersect, or replace (from) the builder's object. `Mode.PRIVATE` instructs the builder that this object should not be combined with the builder's object in any way.

Most commonly, the default `mode` is `Mode.ADD` but this isn't always true. For example, the `Hole` classes use a default `Mode.SUBTRACT` as they remove a volume from the part under normal circumstances. However, the `mode` used in the `Hole` classes can be specified as `Mode.ADD` or `Mode.INTERSECT` to help in inspection or debugging.

1.4.8 Using Locations & Rotating Objects

build123d stores points (to be specific `Location` (s)) internally to be used as positions for the placement of new objects. By default, a single location will be created at the origin of the given workplane such that:

```
with BuildPart() as pipes:
    Box(10, 10, 10, rotation=(10, 20, 30))
```

will create a single 10x10x10 box centered at (0,0,0) - by default objects are centered. One can create multiple objects by pushing points prior to creating objects as follows:

```
with BuildPart() as pipes:
    with Locations((-10, -10, -10), (10, 10, 10)):
        Box(10, 10, 10, rotation=(10, 20, 30))
```

which will create two boxes.

To orient a part, a `rotation` parameter is available on `BuildSketch`` and `BuildPart` APIs. When working in a sketch, the rotation is a single angle in degrees so the parameter is a float. When working on a part, the rotation is a three dimensional `Rotation` object of the form `Rotation(<about x>, <about y>, <about z>)` although a simple three tuple of floats can be used as input. As 3D rotations are not cumulative, one can combine rotations with the * operator like this: `Rotation(10, 20, 30) * Rotation(0, 90, 0)` to generate any desired rotation.

Hint

Experts Only

`Locations` will accept `Location` objects for input which allows one to specify both the position and orientation. However, the orientation is often determined by the `Plane` that an object was created on. `Rotation` is a subclass of `Location` and therefore will also accept a position component.

1.4.9 Builder's Pending Objects

When a builder exits, it will push the object created back to its parent if there was one. Here is an example:

```
height, width, thickness, f_rad = 60, 80, 20, 10

with BuildPart() as pillow_block:
    with BuildSketch() as plan:
        Rectangle(width, height)
        fillet(plan.vertices(), radius=f_rad)
    extrude(amount=thickness)
```

`BuildSketch` exits after the `fillet` operation and when doing so it transfers the sketch to the `pillow_block` instance of `BuildPart` as the internal instance variable `pending_faces`. This allows the `extrude` operation to be immediately invoked as it extrudes these pending faces into `Solid` objects. Likewise, `loft` would take all of the `pending_faces` and attempt to create a single `Solid` object from them.

Normally the user will not need to interact directly with pending objects; however, one can see pending Edges and Faces with `<builder_instance>.pending_edges` and `<builder_instance>.pending_faces` attributes. In the above example, by adding a `print(pillow_block.pending_faces)` prior to the `extrude(amount=thickness)` the pending `Face` from the `BuildSketch` will be displayed.

1.5 Key Concepts (algebra mode)

Build123d's algebra mode works on objects of the classes `Shape`, `Part`, `Sketch` and `Curve` and is based on two concepts:

1. **Object arithmetic**
2. **Placement arithmetic**

1.5.1 Object arithmetic

- Creating a box and a cylinder centered at `(0, 0, 0)`

```
b = Box(1, 2, 3)
c = Cylinder(0.2, 5)
```

- Fusing a box and a cylinder

```
r = Box(1, 2, 3) + Cylinder(0.2, 5)
```

- Cutting a cylinder from a box

```
r = Box(1, 2, 3) - Cylinder(0.2, 5)
```

- Intersecting a box and a cylinder

```
r = Box(1, 2, 3) & Cylinder(0.2, 5)
```

Notes:

- *b*, *c* and *r* are instances of class `Compound` and can be viewed with every viewer that can show `build123d.Compound` objects.
- A discussion around performance can be found in *Performance considerations in algebra mode*.
- A mathematically formal definition of the algebra can be found in *Algebraic definition*.

1.5.2 Placement arithmetic

A `Part`, `Sketch` or `Curve` does not have any location or rotation parameter. The rationale is that an object defines its topology (shape, sizes and its center), but does not know where in space it will be located. Instead, it will be relocated with the `*` operator onto a plane and to location relative to the plane (similar `moved`).

The generic forms of object placement are:

1. Placement on `plane` or at `location` relative to XY plane:

```
plane * alg_compound
location * alg_compound
```

2. Placement on the `plane` and then moved relative to the `plane` by `location` (the location is relative to the local coordinate system of the plane).

```
plane * location * alg_compound
```

Details can be found in *Location arithmetic for algebra mode*.

Examples:

- Box on the `XY` plane, centered at *(0, 0, 0)* (both forms are equivalent):

```
Plane.XY * Box(1, 2, 3)

Box(1, 2, 3)
```

Note: On the `XY` plane no placement is needed (mathematically `Plane.XY *` will not change the location of an object).

- Box on the `XY` plane centered at *(0, 1, 0)* (all three are equivalent):

```
Plane.XY * Pos(0, 1, 0) * Box(1, 2, 3)

Pos(0, 1, 0) * Box(1, 2, 3)

Pos(Y=1) * Box(1, 2, 3)
```

Note: Again, `Plane.XY` can be omitted.

- Box on plane `Plane.XZ`:

```
Plane.XZ * Box(1, 2, 3)
```

- Box on plane `Plane.XZ` with a location `(X=1, Y=2, Z=3)` relative to the `XZ` plane, i.e., using the x-, y- and z-axis of the `XZ` plane:

```
Plane.XZ * Pos(1, 2, 3) * Box(1, 2, 3)
```

- Box on plane `Plane.XZ` moved to `(X=1, Y=2, Z=3)` relative to this plane and rotated there by the angles *(X=0, Y=100, Z=45)* around `Plane.XZ` axes:

```
Plane.XZ * Pos(1, 2, 3) * Rot(0, 100, 45) * Box(1, 2, 3)

Location((1, 2, 3), (0, 100, 45)) * Box(1, 2, 3)
```

Note: `Pos * Rot` is the same as using `Location` directly

- Box on plane `Plane.XZ` rotated on this plane by the angles `(X=0, Y=100, Z=45)` (using the x-, y- and z-axis of the `XZ` plane) and then moved to `(X=1, Y=2, Z=3)` relative to the `XZ` plane:

```
Plane.XZ * Rot(0, 100, 45) * Pos(0,1,2) * Box(1, 2, 3)
```

1.5.3 Combing both concepts

Object arithmetic and **Placement at locations** can be combined:

```
b = Plane.XZ * Rot(X=30) * Box(1, 2, 3) + Plane.YZ * Pos(X=-1) * Cylinder(0.2,␣
↪5)
```

Note: In Python * binds stronger then +, -, &, hence brackets are not needed.

1.6 Moving Objects

In build123d, there are several methods to move objects. These methods vary based on the mode of operation and provide flexibility for object placement and orientation. Below, we outline the three main approaches to moving objects: builder mode, algebra mode, and direct manipulation methods.

1.6.1 Builder Mode

In builder mode, object locations are defined before the objects themselves are created. This approach ensures that objects are positioned correctly during the construction process. The following tools are commonly used to specify locations:

1. *Locations* Use this to define a specific location for the objects within the *with* block.
2. *GridLocations* Arrange objects in a grid pattern.
3. *PolarLocations* Position objects in a circular pattern.
4. *HexLocations* Arrange objects in a hexagonal grid.

> **Note**
>
> The location(s) of an object must be defined prior to its creation when using builder mode.

Example:

```
with Locations((10, 20, 30)):
    Box(5, 5, 5)
```

1.6.2 Algebra Mode

In algebra mode, object movement is expressed using algebraic operations. The *Pos* function, short for Position, represents a location, which can be combined with objects or planes to define placement.

1. `Pos() * shape`: Applies a position to a shape.
2. `Plane() * Pos() * shape`: Combines a plane with a position and applies it to a shape.

Rotation is an important concept in this mode. A *Rotation* represents a location with orientation values set, which can be used to define a new location or modify an existing one.

Example:

```
rotated_box = Rotation(45, 0, 0) * box
```

1.6.3 Direct Manipulation Methods

The following methods allow for direct manipulation of a shape's location and orientation after it has been created. These methods offer a mix of absolute and relative transformations.

Position

- **Absolute Position:** Set the position directly.

```
shape.position = (x, y, z)
```

- **Relative Position:** Adjust the position incrementally.

```
shape.position += (x, y, z)
shape.position -= (x, y, z)
```

Orientation

- **Absolute Orientation:** Set the orientation directly.

```
shape.orientation = (X, Y, Z)
```

- **Relative Orientation:** Adjust the orientation incrementally.

```
shape.orientation += (X, Y, Z)
shape.orientation -= (X, Y, Z)
```

Movement Methods

- **Relative Move:**

```
shape.move(Location)
```

- **Relative Move of Copy:**

```
relocated_shape = shape.moved(Location)
```

- **Absolute Move:**

```
shape.locate(Location)
```

- **Absolute Move of Copy:**

```
relocated_shape = shape.located(Location)
```

Transformation a.k.a. Translation and Rotation

> **Note**
>
> These methods have an optional `transform` parameter which allows the user to transform the base object itself which is quite slow and potentially problematic as opposed to just changing the object's internal *Location*.

- **Translation:** Move a shape relative to its current position.

```
relocated_shape = shape.translate((x, y, z))
```

- **Rotation:** Rotate a shape around a specified axis by a given angle.

```
rotated_shape = shape.rotate(Axis, angle_in_degrees)
```

1.7 Transitioning from OpenSCAD

Welcome to build123d! If you're familiar with OpenSCAD, you'll notice key differences in how models are constructed. This guide is designed to help you adapt your design approach and understand the fundamental differences in modeling philosophies. While OpenSCAD relies heavily on Constructive Solid Geometry (CSG) to combine primitive 3D shapes like cubes and spheres, build123d encourages a more flexible and efficient workflow based on building lower-dimensional objects.

1.7.1 Why Transition to build123d?

Transitioning to build123d allows you to harness a modern and efficient approach to 3D modeling. By starting with lower-dimensional objects and leveraging powerful transformation tools, you can create precise, complex designs with ease. This workflow emphasizes modularity and maintainability, enabling quick modifications and reducing computational complexity.

1.7.2 Moving Beyond Constructive Solid Geometry (CSG)

OpenSCAD's modeling paradigm heavily relies on Constructive Solid Geometry (CSG) to build models by combining and subtracting 3D solids. While build123d supports similar operations, its design philosophy encourages a fundamentally different, often more efficient approach: starting with lower-dimensional entities like faces and edges and then transforming them into solids.

Why Transition Away from CSG?

CSG is a powerful method for creating 3D models, but it has limitations when dealing with complex designs. build123d's approach offers several advantages:

- **Simplified Complexity Management**: Working with 2D profiles and faces instead of directly manipulating 3D solids simplifies your workflow. In large models, the number of operations on solids can grow exponentially, making it difficult to manage and debug. Building with 2D profiles helps keep designs modular and organized.
- **Improved Robustness**: Operations on 2D profiles are inherently less computationally intensive and less error-prone than equivalent operations on 3D solids. This robustness ensures smoother workflows and reduces the likelihood of failing operations in complex models.
- **Enhanced Efficiency**: Constructing models from 2D profiles using operations like **extruding**, **lofting**, **sweeping**, or **revolving** is computationally faster. These methods also provide greater design flexibility, enabling you to create intricate forms with ease.
- **Better Precision and Control**: Starting with 2D profiles allows for more precise geometric control. Constraints, dimensions, and relationships between entities can be established more effectively in 2D, ensuring a solid foundation for your 3D design.

1.7.3 Using a More Traditional CAD Design Workflow

Most industry-standard CAD packages recommend starting with a sketch (a 2D object) and transforming it into a 3D model—a design philosophy that is central to build123d.

In build123d, the design process typically begins with defining the outline of an object. This might involve creating a complex 1D object using **BuildLine**, which provides tools for constructing intricate wireframe geometries. The next step involves converting these 1D objects into 2D sketches using **BuildSketch**, which offers a wide range of 2D primitives and advanced capabilities, such as:

- **make_face**: Converts a 1D **BuildLine** object into a planar 2D face.
- **make_hull**: Generates a convex hull from a 1D **BuildLine** object.

Once a 2D profile is created, it can be transformed into 3D objects in a **BuildPart** context using operations such as:

- **Extrusion**: Extends a 2D profile along a straight path to create a 3D shape.
- **Revolution**: Rotates a 2D profile around an axis to form a symmetrical 3D object.
- **Lofting**: Connects multiple 2D profiles along a path to create smooth transitions between shapes.
- **Sweeping**: Moves a 2D profile along a defined path to create a 3D form.

Refining the Model

After creating the initial 3D shape, you can refine the model by adding details or making modifications using build123d's advanced features, such as:

- **Fillets and Chamfers**: Smooth or bevel edges to enhance the design.
- **Boolean Operations**: Combine, subtract, or intersect 3D shapes to achieve the desired geometry.

Example Comparison

To illustrate the advantages of this approach, compare a simple model in OpenSCAD and build123d of a piece of angle iron:

OpenSCAD Approach

```
$fn = 100; // Increase the resolution for smooth fillets

// Dimensions
length = 100;  // 10 cm long
width = 30;    // 3 cm wide
thickness = 4; // 4 mm thick
fillet = 5;    // 5 mm fillet radius
delta = 0.001; // a small number

// Create the angle iron
difference() {
    // Outer shape
    cube([width, length, width], center = false);
    // Inner shape
    union() {
        translate([thickness+fillet,-delta,thickness+fillet])
            rotate([-90,0,0])
                cylinder(length+2*delta, fillet,fillet);
        translate([thickness,-delta,thickness+fillet])
            cube([width-thickness,length+2*delta,width-fillet],center=false);
        translate([thickness+fillet,-delta,thickness])
            cube([width-fillet,length+2*delta,width-thickness],center=false);

    }
}
```

build123d Approach

```
# Builder mode
with BuildPart() as angle_iron:
    with BuildSketch() as profile:
        Rectangle(3 * CM, 4 * MM, align=Align.MIN)
        Rectangle(4 * MM, 3 * CM, align=Align.MIN)
    extrude(amount=10 * CM)
    fillet(angle_iron.edges().filter_by(lambda e: e.is_interior), 5 * MM)
```

```
# Algebra mode
profile = Rectangle(3 * CM, 4 * MM, align=Align.MIN)
profile += Rectangle(4 * MM, 3 * CM, align=Align.MIN)
```

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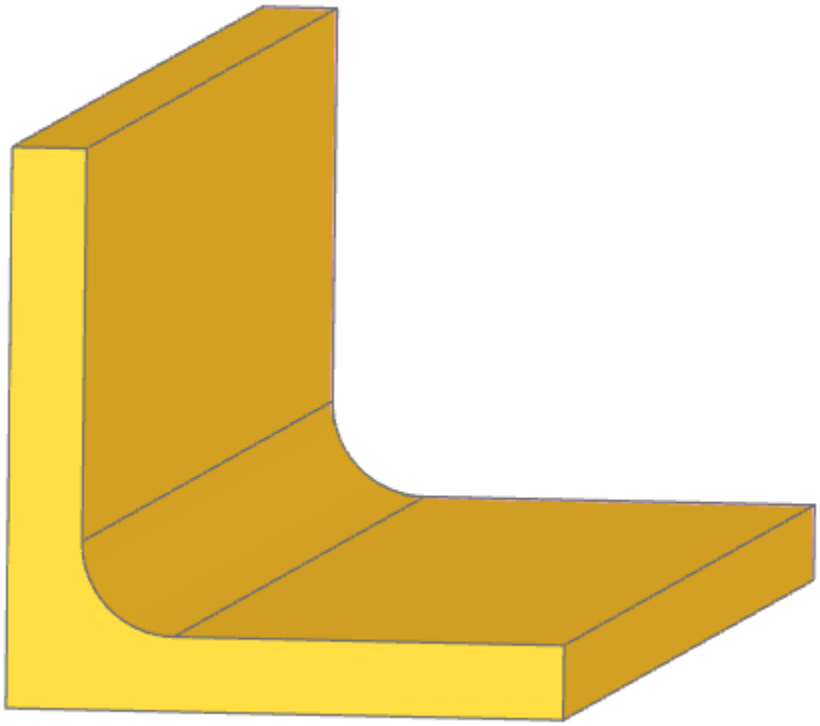
```
angle_iron = extrude(profile, 10 * CM)
angle_iron = fillet(angle_iron.edges().filter_by(lambda e: e.is_interior), 5 * MM)
```



OpenSCAD and build123d offer distinct paradigms for creating 3D models, as demonstrated by the angle iron example. OpenSCAD relies on Constructive Solid Geometry (CSG) operations, combining and subtracting 3D shapes like cubes and cylinders. Fillets are approximated by manually adding high-resolution cylinders, making adjustments cumbersome and less precise. This static approach can handle simple models but becomes challenging for complex or iterative designs.

In contrast, build123d emphasizes a profile-driven workflow. It starts with a 2D sketch, defining the geometry's outline, which is then extruded or otherwise transformed into a 3D model. Features like fillets are applied dynamically by querying topological elements, such as edges, using intuitive filtering methods. This approach ensures precision and flexibility, making changes straightforward without the need for manual repositioning or realignment.

The build123d methodology is computationally efficient, leveraging mathematical precision for features like fillets. By separating the design into manageable steps—sketching, extruding, and refining—it aligns with traditional CAD practices and enhances readability, modularity, and maintainability. Unlike OpenSCAD, build123d's dynamic querying of topological features allows for easy updates and adjustments, making it better suited for modern, complex, and iterative design workflows.

In summary, build123d's sketch-based paradigm and topological querying capabilities provide superior precision, flexibility, and efficiency compared to OpenSCAD's static, CSG-centric approach, making it a better choice for robust and adaptable CAD modeling.

1.7.4 Tips for Transitioning

- **Think in Lower Dimensions**: Begin with 1D curves or 2D sketches as the foundation and progressively build upwards into 3D shapes.
- **Leverage Topological References**: Use build123d's powerful selector system to reference features of existing objects for creating new ones. For example, apply inside or outside fillets and chamfers to vertices and edges of an existing part with precision.
- **Operational Equivalency and Beyond**: Build123d provides equivalents to almost all features available in OpenSCAD, with the exception of the 3D **minkowski** operation. However, a 2D equivalent, **make_hull**, is available in build123d. Beyond operational equivalency, build123d offers a wealth of additional functionality, including advanced features like topological queries, dynamic filtering, and robust tools for creating complex geometries. By exploring build123d's extensive operations, you can unlock new possibilities and take your designs far beyond the capabilities of OpenSCAD.
- **Explore the Documentation**: Dive into build123d's comprehensive API documentation to unlock its full potential and discover advanced features.

By shifting your design mindset from solid-based CSG to a profile-driven approach, you can fully harness build123d's capabilities to create precise, efficient, and complex models. Welcome aboard, and happy designing!

1.7.5 Conclusion

While OpenSCAD and build123d share the goal of empowering users to create parametric 3D models, their approaches differ significantly. Embracing build123d's workflow of building with lower-dimensional objects and applying extrusion, lofting, sweeping, or revolution will unlock its full potential and lead to better design outcomes.

1.8 Introductory Examples

The examples on this page can help you learn how to build objects with build123d, and are intended as a general overview of build123d.

They are organized from simple to complex, so working through them in order is the best way to absorb them.

Note

Some important lines are omitted below to save space, so you will most likely need to add 1 & 2 to the provided code below for them to work:

1. `from build123d import *`
2. If you are using build123d *builder mode* or *algebra mode*,
   - in *ocp_vscode* simply use e.g. `show(ex15)` to the end of your design to view parts, sketches and curves. `show_all()` can be used to automatically show all objects with their variable names as labels.
   - in *CQ-editor* add e.g. `show_object(ex15.part)`, `show_object(ex15.sketch)` or `show_object(ex15.line)` to the end of your design to view parts, sketches or lines.
3. If you want to save your resulting object as an STL from *builder mode*, you can use e.g. `export_stl(ex15.part, "file.stl")`.
4. If you want to save your resulting object as an STL from *algebra mode*, you can use e.g. `export_stl(ex15, "file.stl")`
5. build123d also supports exporting to multiple other file formats including STEP, see here for further information: Import/Export Formats

List of Examples

1.8.1 1. Simple Rectangular Plate

Just about the simplest possible example, a rectangular *Box*.

- **Builder mode**

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex1:
    Box(length, width, thickness)
```

- **Algebra mode**

```
length, width, thickness = 80.0, 60.0, 10.0

ex1 = Box(length, width, thickness)
```

1.8.2 2. Plate with Hole

A rectangular box, but with a hole added.

- **Builder mode**

  In this case we are using *Mode* .SUBTRACT to cut the *Cylinder* from the *Box*.

```
length, width, thickness = 80.0, 60.0, 10.0
center_hole_dia = 22.0

with BuildPart() as ex2:
    Box(length, width, thickness)
    Cylinder(radius=center_hole_dia / 2, height=thickness, mode=Mode.
↪SUBTRACT)
```

- **Algebra mode**

  In this case we are using the subtract operator - to cut the *Cylinder* from the *Box*.

```
length, width, thickness = 80.0, 60.0, 10.0
center_hole_dia = 22.0

ex2 = Box(length, width, thickness)
ex2 -= Cylinder(center_hole_dia / 2, height=thickness)
```

1.8.3 3. An extruded prismatic solid

Build a prismatic solid using extrusion.

- **Builder mode**

  This time we can first create a 2D *BuildSketch* adding a *Circle* and a subtracted *Rectangle* and then use *BuildPart*'s *extrude()* feature.

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex3:
    with BuildSketch() as ex3_sk:
        Circle(width)
        Rectangle(length / 2, width / 2, mode=Mode.SUBTRACT)
    extrude(amount=2 * thickness)
```

- **Algebra mode**

  This time we can first create a 2D *Circle* with a subtracted `Rectangle` and then use the *extrude()* operation for parts.

```
length, width, thickness = 80.0, 60.0, 10.0

sk3 = Circle(width) - Rectangle(length / 2, width / 2)
ex3 = extrude(sk3, amount=2 * thickness)
```

1.8.4 4. Building Profiles using lines and arcs

Sometimes you need to build complex profiles using lines and arcs. This example builds a prismatic solid from 2D operations. It is not necessary to create variables for the line segments, but it will be useful in a later example.

- **Builder mode**

  BuildSketch operates on closed Faces, and the operation *make_face()* is used to convert the pending line segments from *BuildLine* into a closed Face.

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex4:
    with BuildSketch() as ex4_sk:
        with BuildLine() as ex4_ln:
            l1 = Line((0, 0), (length, 0))
            l2 = Line((length, 0), (length, width))
            l3 = ThreePointArc((length, width), (width, width * 1.5), (0.0,␣
↪width))
            l4 = Line((0.0, width), (0, 0))
        make_face()
    extrude(amount=thickness)
```

- **Algebra mode**

  We start with an empty *Curve* and add lines to it (note that `Curve() + [line1, line2, line3]` is much more efficient than `line1 + line2 + line3`, see *Performance considerations in algebra mode*). The operation *make_face()* is used to convert the line segments into a Face.

```
length, width, thickness = 80.0, 60.0, 10.0

lines = Curve() + [
    Line((0, 0), (length, 0)),
    Line((length, 0), (length, width)),
    ThreePointArc((length, width), (width, width * 1.5), (0.0, width)),
    Line((0.0, width), (0, 0)),
]
sk4 = make_face(lines)
ex4 = extrude(sk4, thickness)
```

Note that to build a closed face it requires line segments that form a closed shape.

1.8.5 5. Moving the current working point

- **Builder mode**

  Using *Locations* we can place one (or multiple) objects at one (or multiple) places.

```
a, b, c, d = 90, 45, 15, 7.5

with BuildPart() as ex5:
    with BuildSketch() as ex5_sk:
        Circle(a)
        with Locations((b, 0.0)):
            Rectangle(c, c, mode=Mode.SUBTRACT)
        with Locations((0, b)):
            Circle(d, mode=Mode.SUBTRACT)
    extrude(amount=c)
```

- **Algebra mode**

  Using the pattern `Pos(x, y, z=0) * obj` (with *geometry.Pos*) we can move an object to the provided position. Using `Rot(x_angle, y_angle, z_angle) * obj` (with *geometry.Rot*) would rotate the object.

```
a, b, c, d = 90, 45, 15, 7.5

sk5 = Circle(a) - Pos(b, 0.0) * Rectangle(c, c) - Pos(0.0, b) * Circle(d)
ex5 = extrude(sk5, c)
```

1.8.6 6. Using Point Lists

Sometimes you need to create a number of features at various *Locations*.

- **Builder mode**

  You can use a list of points to construct multiple objects at once.

```
a, b, c = 80, 60, 10

with BuildPart() as ex6:
```

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```
    with BuildSketch() as ex6_sk:
        Circle(a)
        with Locations((b, 0), (0, b), (-b, 0), (0, -b)):
            Circle(c, mode=Mode.SUBTRACT)
    extrude(amount=c)
```

- **Algebra mode**

  You can use loops to iterate over these Locations or list comprehensions as in the example.

  The algebra operations are vectorized, which means `obj - [obj1, obj2, obj3]` is short for `obj - obj1 - obj2 - ob3` (and more efficient, see *Performance considerations in algebra mode*).

```
a, b, c = 80, 60, 10

sk6 = [loc * Circle(c) for loc in Locations((b, 0), (0, b), (-b, 0), (0, -
↪b))]
ex6 = extrude(Circle(a) - sk6, c)
```

1.8.7 7. Polygons

- **Builder mode**

  You can create *RegularPolygon* for each stack point if you would like.

```
a, b, c = 60, 80, 5

with BuildPart() as ex7:
    with BuildSketch() as ex7_sk:
        Rectangle(a, b)
        with Locations((0, 3 * c), (0, -3 * c)):
            RegularPolygon(radius=2 * c, side_count=6, mode=Mode.SUBTRACT)
    extrude(amount=c)
```

- **Algebra mode**

  You can apply locations to *RegularPolygon* instances for each location via loops or list comprehensions.

```
a, b, c = 60, 80, 5

polygons = [
    loc * RegularPolygon(radius=2 * c, side_count=6)
    for loc in Locations((0, 3 * c), (0, -3 * c))
]
sk7 = Rectangle(a, b) - polygons
ex7 = extrude(sk7, amount=c)
```

1.8.8 8. Polylines

Polyline allows creating a shape from a large number of chained points connected by lines. This example uses a polyline to create one half of an i-beam shape, which is *mirror()* ed to create the final profile.

- **Builder mode**

```
(L, H, W, t) = (100.0, 20.0, 20.0, 1.0)
pts = [
    (0, H / 2.0),
    (W / 2.0, H / 2.0),
    (W / 2.0, (H / 2.0 - t)),
    (t / 2.0, (H / 2.0 - t)),
    (t / 2.0, (t - H / 2.0)),
    (W / 2.0, (t - H / 2.0)),
    (W / 2.0, H / -2.0),
    (0, H / -2.0),
]

with BuildPart() as ex8:
    with BuildSketch(Plane.YZ) as ex8_sk:
        with BuildLine() as ex8_ln:
            Polyline(pts)
            mirror(ex8_ln.line, about=Plane.YZ)
        make_face()
    extrude(amount=L)
```

- **Algebra mode**

```
(L, H, W, t) = (100.0, 20.0, 20.0, 1.0)
pts = [
    (0, H / 2.0),
    (W / 2.0, H / 2.0),
    (W / 2.0, (H / 2.0 - t)),
    (t / 2.0, (H / 2.0 - t)),
    (t / 2.0, (t - H / 2.0)),
    (W / 2.0, (t - H / 2.0)),
    (W / 2.0, H / -2.0),
    (0, H / -2.0),
]

ln = Polyline(pts)
ln += mirror(ln, Plane.YZ)

sk8 = make_face(Plane.YZ * ln)
ex8 = extrude(sk8, -L).clean()
```

1.8.9 9. Selectors, Fillets, and Chamfers

This example introduces multiple useful and important concepts. Firstly *chamfer()* and *fillet()* can be used to "bevel" and "round" edges respectively. Secondly, these two methods require an edge or a list of edges to operate on. To select all edges, you could simply pass in `ex9.edges()`.

- **Builder mode**

```
length, width, thickness = 80.0, 60.0, 10.0
```

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```
with BuildPart() as ex9:
    Box(length, width, thickness)
    chamfer(ex9.edges().group_by(Axis.Z)[-1], length=4)
    fillet(ex9.edges().filter_by(Axis.Z), radius=5)
```

- **Algebra mode**

```
length, width, thickness = 80.0, 60.0, 10.0

ex9 = Part() + Box(length, width, thickness)
ex9 = chamfer(ex9.edges().group_by(Axis.Z)[-1], length=4)
ex9 = fillet(ex9.edges().filter_by(Axis.Z), radius=5)
```

Note that *group_by()* (Axis.Z) returns a list of lists of edges that is grouped by their z-position. In this case we want to use the [-1] group which, by convention, will be the highest z-dimension group.

1.8.10 10. Select Last and Hole

- **Builder mode**

  Using *Select* .LAST you can select the most recently modified edges. It is used to perform a *fillet()* in this example. This example also makes use of *Hole* which automatically cuts through the entire part.

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex10:
    Box(length, width, thickness)
    Hole(radius=width / 4)
    fillet(ex10.edges(Select.LAST).group_by(Axis.Z)[-1], radius=2)
```

- **Algebra mode**

  Using the pattern snapshot = obj.edges() before and last_edges = obj.edges() - snapshot after an operation allows to select the most recently modified edges (same for faces, vertices, ...). It is used to perform a *fillet()* in this example. This example also makes use of *Hole*. Different to the *context mode*, you have to add the depth of the whole.

```
ex10 = Part() + Box(length, width, thickness)

snapshot = ex10.edges()
ex10 -= Hole(radius=width / 4, depth=thickness)
last_edges = ex10.edges() - snapshot
ex10 = fillet(last_edges.group_by(Axis.Z)[-1], 2)
```

1.8.11 11. Use a face as a plane for BuildSketch and introduce GridLocations

- **Builder mode**

  BuildSketch accepts a Plane or a Face, so in this case we locate the Sketch on the top of the part. Note that the face used as input to BuildSketch needs to be Planar or unpredictable behavior can result.

Additionally *GridLocations* can be used to create a grid of points that are simultaneously used to place 4 pentagons.

Lastly, *extrude()* can be used with a negative amount and `Mode.SUBTRACT` to cut these from the parent.

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex11:
    Box(length, width, thickness)
    chamfer(ex11.edges().group_by(Axis.Z)[-1], length=4)
    fillet(ex11.edges().filter_by(Axis.Z), radius=5)
    Hole(radius=width / 4)
    fillet(ex11.edges(Select.LAST).sort_by(Axis.Z)[-1], radius=2)
    with BuildSketch(ex11.faces().sort_by(Axis.Z)[-1]) as ex11_sk:
        with GridLocations(length / 2, width / 2, 2, 2):
            RegularPolygon(radius=5, side_count=5)
    extrude(amount=-thickness, mode=Mode.SUBTRACT)
```

- **Algebra mode**

  The pattern `plane * obj` can be used to locate an object on a plane. Furthermore, the pattern `plane * location * obj` first places the object on a plane and then moves it relative to plane according to `location`.

  GridLocations creates a grid of points that can be used in loops or list comprehensions as described earlier.

  Lastly, *extrude()* can be used with a negative amount and cut (`-`) from the parent.

```
length, width, thickness = 80.0, 60.0, 10.0

ex11 = Part() + Box(length, width, thickness)
ex11 = chamfer(ex11.edges().group_by()[-1], 4)
ex11 = fillet(ex11.edges().filter_by(Axis.Z), 5)
last = ex11.edges()
ex11 -= Hole(radius=width / 4, depth=thickness)
ex11 = fillet((ex11.edges() - last).sort_by().last, 2)

plane = Plane(ex11.faces().sort_by().last)
polygons = Sketch() + [
    plane * loc * RegularPolygon(radius=5, side_count=5)
    for loc in GridLocations(length / 2, width / 2, 2, 2)
]
ex11 -= extrude(polygons, -thickness)
```

Note that the direction implied by positive or negative inputs to amount is relative to the normal direction of the face or plane. As a result of this, unexpected behavior can occur if the extrude direction and mode/operation (ADD / `+` or SUBTRACT / `-`) are not correctly set.

1.8.12 12. Defining an Edge with a Spline

This example defines a side using a spline curve through a collection of points. Useful when you have an edge that needs a complex profile.

- **Builder mode**

```
pts = [
    (55, 30),
    (50, 35),
    (40, 30),
    (30, 20),
    (20, 25),
    (10, 20),
    (0, 20),
]

with BuildPart() as ex12:
    with BuildSketch() as ex12_sk:
        with BuildLine() as ex12_ln:
            l1 = Spline(pts)
            l2 = Line((55, 30), (60, 0))
            l3 = Line((60, 0), (0, 0))
            l4 = Line((0, 0), (0, 20))
        make_face()
    extrude(amount=10)
```

- **Algebra mode**

```
pts = [
    (55, 30),
    (50, 35),
    (40, 30),
    (30, 20),
    (20, 25),
    (10, 20),
    (0, 20),
]

l1 = Spline(pts)
l2 = Line(l1 @ 0, (60, 0))
l3 = Line(l2 @ 1, (0, 0))
l4 = Line(l3 @ 1, l1 @ 1)

sk12 = make_face([l1, l2, l3, l4])
ex12 = extrude(sk12, 10)
```

1.8.13 13. CounterBoreHoles, CounterSinkHoles, and PolarLocations

Counter-sink and counter-bore holes are useful for creating recessed areas for fasteners.

- **Builder mode**

  We use a face to establish a location for *Locations*.

```
a, b = 40, 4
with BuildPart() as ex13:
    Cylinder(radius=50, height=10)
    with Locations(ex13.faces().sort_by(Axis.Z)[-1]):
```

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```
        with PolarLocations(radius=a, count=4):
            CounterSinkHole(radius=b, counter_sink_radius=2 * b)
        with PolarLocations(radius=a, count=4, start_angle=45, angular_
↪range=360):
            CounterBoreHole(radius=b, counter_bore_radius=2 * b, counter_
↪bore_depth=b)
```

- **Algebra mode**

  We use a face to establish a plane that is used later in the code for locating objects onto this plane.

```
a, b = 40, 4

ex13 = Cylinder(radius=50, height=10)
plane = Plane(ex13.faces().sort_by().last)

ex13 -= (
    plane
    * PolarLocations(radius=a, count=4)
    * CounterSinkHole(radius=b, counter_sink_radius=2 * b, depth=10)
)
ex13 -= (
    plane
    * PolarLocations(radius=a, count=4, start_angle=45, angular_range=360)
    * CounterBoreHole(
        radius=b, counter_bore_radius=2 * b, depth=10, counter_bore_depth=b
    )
)
```

PolarLocations creates a list of points that are radially distributed.

1.8.14 14. Position on a line with '@', '%' and introduce Sweep

build123d includes a feature for finding the position along a line segment. This is normalized between 0 and 1 and can be accessed using the *position_at()* (@) operator. Similarly the *tangent_at()* (%) operator returns the line direction at a given point.

These two features are very powerful for chaining line segments together without having to repeat dimensions again and again, which is error prone, time consuming, and more difficult to maintain. The pending faces must lie on the path, please see example 37 for a way to make this placement easier.

- **Builder mode**

  The *sweep()* method takes any pending faces and sweeps them through the provided path (in this case the path is taken from the pending edges from `ex14_ln`). *revolve()* requires a single connected wire.

```
a, b = 40, 20

with BuildPart() as ex14:
    with BuildLine() as ex14_ln:
        l1 = JernArc(start=(0, 0), tangent=(0, 1), radius=a, arc_size=180)
        l2 = JernArc(start=l1 @ 1, tangent=l1 % 1, radius=a, arc_size=-90)
```

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```
        l3 = Line(l2 @ 1, l2 @ 1 + (-a, a))
    with BuildSketch(Plane.XZ) as ex14_sk:
        Rectangle(b, b)
    sweep()
```

- **Algebra mode**

  The *sweep()* method takes any faces and sweeps them through the provided path (in this case the path is taken from `ex14_ln`).

```
a, b = 40, 20

l1 = JernArc(start=(0, 0), tangent=(0, 1), radius=a, arc_size=180)
l2 = JernArc(start=l1 @ 1, tangent=l1 % 1, radius=a, arc_size=-90)
l3 = Line(l2 @ 1, l2 @ 1 + (-a, a))
ex14_ln = l1 + l2 + l3

sk14 = Plane.XZ * Rectangle(b, b)
ex14 = sweep(sk14, path=ex14_ln)
```

It is also possible to use tuple or *Vector* addition (and other vector math operations) as seen in the `l3` variable.

1.8.15 15. Mirroring Symmetric Geometry

Here mirror is used on the BuildLine to create a symmetric shape with fewer line segment commands. Additionally the '@' operator is used to simplify the line segment commands.

`(l4 @ 1).Y` is used to extract the y-component of the `l4 @ 1` vector.

- **Builder mode**

```
a, b, c = 80, 40, 20

with BuildPart() as ex15:
    with BuildSketch() as ex15_sk:
        with BuildLine() as ex15_ln:
            l1 = Line((0, 0), (a, 0))
            l2 = Line(l1 @ 1, l1 @ 1 + (0, b))
            l3 = Line(l2 @ 1, l2 @ 1 + (-c, 0))
            l4 = Line(l3 @ 1, l3 @ 1 + (0, -c))
            l5 = Line(l4 @ 1, (0, (l4 @ 1).Y))
            mirror(ex15_ln.line, about=Plane.YZ)
        make_face()
    extrude(amount=c)
```

- **Algebra mode**

  Combine lines via the pattern `Curve() + [l1, l2, l3, l4, l5]`

```
a, b, c = 80, 40, 20

l1 = Line((0, 0), (a, 0))
l2 = Line(l1 @ 1, l1 @ 1 + (0, b))
```

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```
l3 = Line(l2 @ 1, l2 @ 1 + (-c, 0))
l4 = Line(l3 @ 1, l3 @ 1 + (0, -c))
l5 = Line(l4 @ 1, (0, (l4 @ 1).Y))
ln = Curve() + [l1, l2, l3, l4, l5]
ln += mirror(ln, Plane.YZ)

sk15 = make_face(ln)
ex15 = extrude(sk15, c)
```

1.8.16 16. Mirroring 3D Objects

Mirror can also be used with BuildPart (and BuildSketch) to mirror 3D objects. The `Plane.offset()` method shifts the plane in the normal direction (positive or negative).

- **Builder mode**

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex16_single:
    with BuildSketch(Plane.XZ) as ex16_sk:
        Rectangle(length, width)
        fillet(ex16_sk.vertices(), radius=length / 10)
        with GridLocations(x_spacing=length / 4, y_spacing=0, x_count=3, y_
↪count=1):
            Circle(length / 12, mode=Mode.SUBTRACT)
        Rectangle(length, width, align=(Align.MIN, Align.MIN), mode=Mode.
↪SUBTRACT)
    extrude(amount=length)

with BuildPart() as ex16:
    add(ex16_single.part)
    mirror(ex16_single.part, about=Plane.XY.offset(width))
    mirror(ex16_single.part, about=Plane.YX.offset(width))
    mirror(ex16_single.part, about=Plane.YZ.offset(width))
    mirror(ex16_single.part, about=Plane.YZ.offset(-width))
```

- **Algebra mode**

```
length, width, thickness = 80.0, 60.0, 10.0

sk16 = Rectangle(length, width)
sk16 = fillet(sk16.vertices(), length / 10)

circles = [loc * Circle(length / 12) for loc in GridLocations(length / 4, 0,
↪ 3, 1)]

sk16 = sk16 - circles - Rectangle(length, width, align=(Align.MIN, Align.
↪MIN))
ex16_single = extrude(Plane.XZ * sk16, length)

planes = [
```

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```
    Plane.XY.offset(width),
    Plane.YX.offset(width),
    Plane.YZ.offset(width),
    Plane.YZ.offset(-width),
]
objs = [mirror(ex16_single, plane) for plane in planes]
ex16 = ex16_single + objs
```

1.8.17 17. Mirroring From Faces

Here we select the farthest face in the Y-direction and turn it into a *Plane* using the `Plane()` class.

- **Builder mode**

```
a, b = 30, 20

with BuildPart() as ex17:
    with BuildSketch() as ex17_sk:
        RegularPolygon(radius=a, side_count=5)
    extrude(amount=b)
    mirror(ex17.part, about=Plane(ex17.faces().group_by(Axis.Y)[0][0]))
```

- **Algebra mode**

```
a, b = 30, 20

sk17 = RegularPolygon(radius=a, side_count=5)
ex17 = extrude(sk17, amount=b)
ex17 += mirror(ex17, Plane(ex17.faces().sort_by(Axis.Y).first))
```

1.8.18 18. Creating Workplanes on Faces

Here we start with an earlier example, select the top face, draw a rectangle and then use Extrude with a negative distance.

- **Builder mode**

  We then use `Mode.SUBTRACT` to cut it out from the main body.

```
length, width, thickness = 80.0, 60.0, 10.0
a, b = 4, 5

with BuildPart() as ex18:
    Box(length, width, thickness)
    chamfer(ex18.edges().group_by(Axis.Z)[-1], length=a)
    fillet(ex18.edges().filter_by(Axis.Z), radius=b)
    with BuildSketch(ex18.faces().sort_by(Axis.Z)[-1]):
        Rectangle(2 * b, 2 * b)
    extrude(amount=-thickness, mode=Mode.SUBTRACT)
```

- **Algebra mode**

  We then use `-=` to cut it out from the main body.

```
length, width, thickness = 80.0, 60.0, 10.0
a, b = 4, 5

ex18 = Part() + Box(length, width, thickness)
ex18 = chamfer(ex18.edges().group_by()[-1], a)
ex18 = fillet(ex18.edges().filter_by(Axis.Z), b)

sk18 = Plane(ex18.faces().sort_by().first) * Rectangle(2 * b, 2 * b)
ex18 -= extrude(sk18, -thickness)
```

1.8.19 19. Locating a workplane on a vertex

Here a face is selected and two different strategies are used to select vertices. Firstly `vtx` uses *group_by()* and `Axis.X` to select a particular vertex. The second strategy uses a custom defined Axis `vtx2Axis` that is pointing roughly in the direction of a vertex to select, and then *sort_by()* this custom Axis.

- **Builder mode**

  Then the X and Y positions of these vertices are selected and passed to *Locations* as center points for two circles that cut through the main part. Note that if you passed the variable `vtx` directly to *Locations* then the part would be offset from the workplane by the vertex z-position.

```
length, thickness = 80.0, 10.0

with BuildPart() as ex19:
    with BuildSketch() as ex19_sk:
        RegularPolygon(radius=length / 2, side_count=7)
    extrude(amount=thickness)
    topf = ex19.faces().sort_by(Axis.Z)[-1]
    vtx = topf.vertices().group_by(Axis.X)[-1][0]
    vtx2Axis = Axis((0, 0, 0), (-1, -0.5, 0))
    vtx2 = topf.vertices().sort_by(vtx2Axis)[-1]
    with BuildSketch(topf) as ex19_sk2:
        with Locations((vtx.X, vtx.Y), (vtx2.X, vtx2.Y)):
            Circle(radius=length / 8)
    extrude(amount=-thickness, mode=Mode.SUBTRACT)
```

- **Algebra mode**

  Then the X and Y positions of these vertices are selected and used to move two circles that cut through the main part. Note that if you passed the variable `vtx` directly to *Pos* then the part would be offset from the workplane by the vertex z-position.

```
length, thickness = 80.0, 10.0

ex19_sk = RegularPolygon(radius=length / 2, side_count=7)
ex19 = extrude(ex19_sk, thickness)

topf = ex19.faces().sort_by().last

vtx = topf.vertices().group_by(Axis.X)[-1][0]

vtx2Axis = Axis((0, 0, 0), (-1, -0.5, 0))
```

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```
vtx2 = topf.vertices().sort_by(vtx2Axis)[-1]

ex19_sk2 = Circle(radius=length / 8)
ex19_sk2 = Pos(vtx.X, vtx.Y) * ex19_sk2 + Pos(vtx2.X, vtx2.Y) * ex19_sk2

ex19 -= extrude(ex19_sk2, thickness)
```

1.8.20 20. Offset Sketch Workplane

The `plane` variable is set to be coincident with the farthest face in the negative x-direction. The resulting Plane is offset from the original position.

- **Builder mode**

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex20:
    Box(length, width, thickness)
    plane = Plane(ex20.faces().group_by(Axis.X)[0][0])
    with BuildSketch(plane.offset(2 * thickness)):
        Circle(width / 3)
    extrude(amount=width)
```

- **Algebra mode**

```
length, width, thickness = 80.0, 60.0, 10.0

ex20 = Box(length, width, thickness)
plane = Plane(ex20.faces().sort_by(Axis.X).first).offset(2 * thickness)

sk20 = plane * Circle(width / 3)
ex20 += extrude(sk20, width)
```

1.8.21 21. Create a Workplanes in the center of another shape

One cylinder is created, and then the origin and z_dir of that part are used to create a new Plane for positioning another cylinder perpendicular and halfway along the first.

- **Builder mode**

```
width, length = 10.0, 60.0

with BuildPart() as ex21:
    with BuildSketch() as ex21_sk:
        Circle(width / 2)
    extrude(amount=length)
    with BuildSketch(Plane(origin=ex21.part.center(), z_dir=(-1, 0, 0))):
        Circle(width / 2)
    extrude(amount=length)
```

- **Algebra mode**

```
width, length = 10.0, 60.0

ex21 = extrude(Circle(width / 2), length)
plane = Plane(origin=ex21.center(), z_dir=(-1, 0, 0))
ex21 += plane * extrude(Circle(width / 2), length)
```

1.8.22 22. Rotated Workplanes

It is also possible to create a rotated workplane, building upon some of the concepts in an earlier example.

- **Builder mode**

  Use the *rotated()* method to rotate the workplane.

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex22:
    Box(length, width, thickness)
    pln = Plane(ex22.faces().group_by(Axis.Z)[0][0]).rotated((0, -50, 0))
    with BuildSketch(pln) as ex22_sk:
        with GridLocations(length / 4, width / 4, 2, 2):
            Circle(thickness / 4)
    extrude(amount=-100, both=True, mode=Mode.SUBTRACT)
```

- **Algebra mode**

  Use the operator * to relocate the plane (post-multiplication!).

```
length, width, thickness = 80.0, 60.0, 10.0

ex22 = Box(length, width, thickness)
plane = Plane((ex22.faces().group_by(Axis.Z)[0])[0]) * Rot(0, 50, 0)

holes = Sketch() + [
    plane * loc * Circle(thickness / 4)
    for loc in GridLocations(length / 4, width / 4, 2, 2)
]
ex22 -= extrude(holes, -100, both=True)
```

GridLocations places 4 Circles on 4 points on this rotated workplane, and then the Circles are extruded in the "both" (positive and negative) normal direction.

1.8.23 23. Revolve

Here we build a sketch with a *Polyline*, *Line*, and a *Circle*. It is absolutely critical that the sketch is only on one side of the axis of rotation before Revolve is called. To that end, `split` is used with `Plane.ZY` to keep only one side of the Sketch.

It is highly recommended to view your sketch before you attempt to call revolve.

- **Builder mode**

```
pts = [
    (-25, 35),
    (-25, 0),
    (-20, 0),
    (-20, 5),
    (-15, 10),
    (-15, 35),
]

with BuildPart() as ex23:
    with BuildSketch(Plane.XZ) as ex23_sk:
        with BuildLine() as ex23_ln:
            l1 = Polyline(pts)
            l2 = Line(l1 @ 1, l1 @ 0)
        make_face()
        with Locations((0, 35)):
            Circle(25)
        split(bisect_by=Plane.ZY)
    revolve(axis=Axis.Z)
```

- **Algebra mode**

```
pts = [
    (-25, 35),
    (-25, 0),
    (-20, 0),
    (-20, 5),
    (-15, 10),
    (-15, 35),
]

l1 = Polyline(pts)
l2 = Line(l1 @ 1, l1 @ 0)
sk23 = make_face([l1, l2])

sk23 += Pos(0, 35) * Circle(25)
sk23 = Plane.XZ * split(sk23, bisect_by=Plane.ZY)

ex23 = revolve(sk23, Axis.Z)
```

1.8.24 24. Loft

Loft is a very powerful tool that can be used to join dissimilar shapes. In this case we make a conical-like shape from a circle and a rectangle that is offset vertically. In this case *loft()* automatically takes the pending faces that were added by the two BuildSketches. Loft can behave unexpectedly when the input faces are not parallel to each other.

- **Builder mode**

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex24:
    Box(length, length, thickness)
```

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```
    with BuildSketch(ex24.faces().group_by(Axis.Z)[0][0]) as ex24_sk:
        Circle(length / 3)
    with BuildSketch(ex24_sk.faces()[0].offset(length / 2)) as ex24_sk2:
        Rectangle(length / 6, width / 6)
    loft()
```

- **Algebra mode**

```
length, width, thickness = 80.0, 60.0, 10.0

ex24 = Box(length, length, thickness)
plane = Plane(ex24.faces().sort_by().last)

faces = Sketch() + [
    plane * Circle(length / 3),
    plane.offset(length / 2) * Rectangle(length / 6, width / 6),
]

ex24 += loft(faces)
```

1.8.25 25. Offset Sketch

- **Builder mode**

  BuildSketch faces can be transformed with a 2D *offset()*.

```
rad, offs = 50, 10

with BuildPart() as ex25:
    with BuildSketch() as ex25_sk1:
        RegularPolygon(radius=rad, side_count=5)
    with BuildSketch(Plane.XY.offset(15)) as ex25_sk2:
        RegularPolygon(radius=rad, side_count=5)
        offset(amount=offs)
    with BuildSketch(Plane.XY.offset(30)) as ex25_sk3:
        RegularPolygon(radius=rad, side_count=5)
        offset(amount=offs, kind=Kind.INTERSECTION)
    extrude(amount=1)
```

- **Algebra mode**

  Sketch faces can be transformed with a 2D *offset()*.

```
rad, offs = 50, 10

sk25_1 = RegularPolygon(radius=rad, side_count=5)
sk25_2 = Plane.XY.offset(15) * RegularPolygon(radius=rad, side_count=5)
sk25_2 = offset(sk25_2, offs)
sk25_3 = Plane.XY.offset(30) * RegularPolygon(radius=rad, side_count=5)
sk25_3 = offset(sk25_3, offs, kind=Kind.INTERSECTION)
```

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```
sk25 = Sketch() + [sk25_1, sk25_2, sk25_3]
ex25 = extrude(sk25, 1)
```

They can be offset inwards or outwards, and with different techniques for extending the corners (see *Kind* in the Offset docs).

1.8.26 26. Offset Part To Create Thin features

Parts can also be transformed using an offset, but in this case with a 3D *offset()*. Also commonly known as a shell, this allows creating thin walls using very few operations. This can also be offset inwards or outwards. Faces can be selected to be "deleted" using the `openings` parameter of *offset()*.

Note that self intersecting edges and/or faces can break both 2D and 3D offsets.

- **Builder mode**

```
length, width, thickness, wall = 80.0, 60.0, 10.0, 2.0

with BuildPart() as ex26:
    Box(length, width, thickness)
    topf = ex26.faces().sort_by(Axis.Z)[-1]
    offset(amount=-wall, openings=topf)
```

- **Algebra mode**

```
length, width, thickness, wall = 80.0, 60.0, 10.0, 2.0

ex26 = Box(length, width, thickness)
topf = ex26.faces().sort_by().last
ex26 = offset(ex26, amount=-wall, openings=topf)
```

1.8.27 27. Splitting an Object

You can split an object using a plane, and retain either or both halves. In this case we select a face and offset half the width of the box.

- **Builder mode**

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex27:
    Box(length, width, thickness)
    with BuildSketch(ex27.faces().sort_by(Axis.Z)[0]) as ex27_sk:
        Circle(width / 4)
    extrude(amount=-thickness, mode=Mode.SUBTRACT)
    split(bisect_by=Plane(ex27.faces().sort_by(Axis.Y)[-1]).offset(-width /␣
↪2))
```

- **Algebra mode**

```
length, width, thickness = 80.0, 60.0, 10.0

ex27 = Box(length, width, thickness)
sk27 = Plane(ex27.faces().sort_by().first) * Circle(width / 4)
ex27 -= extrude(sk27, -thickness)
ex27 = split(ex27, Plane(ex27.faces().sort_by(Axis.Y).last).offset(-width /␣
↪2))
```

1.8.28 28. Locating features based on Faces

- **Builder mode**

  We create a triangular prism with *Mode* .PRIVATE and then later use the faces of this object to cut holes in a sphere.

```
width, thickness = 80.0, 10.0

with BuildPart() as ex28:
    with BuildSketch() as ex28_sk:
        RegularPolygon(radius=width / 4, side_count=3)
    ex28_ex = extrude(amount=thickness, mode=Mode.PRIVATE)
    midfaces = ex28_ex.faces().group_by(Axis.Z)[1]
    Sphere(radius=width / 2)
    for face in midfaces:
        with Locations(face):
            Hole(thickness / 2)
```

- **Algebra mode**

  We create a triangular prism and then later use the faces of this object to cut holes in a sphere.

```
width, thickness = 80.0, 10.0

sk28 = RegularPolygon(radius=width / 4, side_count=3)
tmp28 = extrude(sk28, thickness)
ex28 = Sphere(radius=width / 2)
for p in [Plane(face) for face in tmp28.faces().group_by(Axis.Z)[1]]:
    ex28 -= p * Hole(thickness / 2, depth=width)
```

We are able to create multiple workplanes by looping over the list of faces.

1.8.29 29. The Classic OCC Bottle

build123d is based on the OpenCascade.org (OCC) modeling Kernel. Those who are familiar with OCC know about the famous 'bottle' example. We use a 3D Offset and the openings parameter to create the bottle opening.

- **Builder mode**

```
L, w, t, b, h, n = 60.0, 18.0, 9.0, 0.9, 90.0, 6.0

with BuildPart() as ex29:
```

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```
    with BuildSketch(Plane.XY.offset(-b)) as ex29_ow_sk:
        with BuildLine() as ex29_ow_ln:
            l1 = Line((0, 0), (0, w / 2))
            l2 = ThreePointArc(l1 @ 1, (L / 2.0, w / 2.0 + t), (L, w / 2.0))
            l3 = Line(l2 @ 1, ((l2 @ 1).X, 0, 0))
            mirror(ex29_ow_ln.line)
        make_face()
    extrude(amount=h + b)
    fillet(ex29.edges(), radius=w / 6)
    with BuildSketch(ex29.faces().sort_by(Axis.Z)[-1]):
        Circle(t)
    extrude(amount=n)
    necktopf = ex29.faces().sort_by(Axis.Z)[-1]
    offset(ex29.solids()[0], amount=-b, openings=necktopf)
```

- **Algebra mode**

```
L, w, t, b, h, n = 60.0, 18.0, 9.0, 0.9, 90.0, 8.0

l1 = Line((0, 0), (0, w / 2))
l2 = ThreePointArc(l1 @ 1, (L / 2.0, w / 2.0 + t), (L, w / 2.0))
l3 = Line(l2 @ 1, ((l2 @ 1).X, 0, 0))
ln29 = l1 + l2 + l3
ln29 += mirror(ln29)
sk29 = make_face(ln29)
ex29 = extrude(sk29, -(h + b))
ex29 = fillet(ex29.edges(), radius=w / 6)

neck = Plane(ex29.faces().sort_by().last) * Circle(t)
ex29 += extrude(neck, n)
necktopf = ex29.faces().sort_by().last
ex29 = offset(ex29, -b, openings=necktopf)
```

1.8.30 30. Bezier Curve

Here `pts` is used as an input to both *Polyline* and *Bezier* and `wts` to Bezier alone. These two together create a closed line that is made into a face and extruded.

- **Builder mode**

```
pts = [
    (0, 0),
    (20, 20),
    (40, 0),
    (0, -40),
    (-60, 0),
    (0, 100),
    (100, 0),
]

wts = [
```

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```
    1.0,
    1.0,
    2.0,
    3.0,
    4.0,
    2.0,
    1.0,
]

with BuildPart() as ex30:
    with BuildSketch() as ex30_sk:
        with BuildLine() as ex30_ln:
            l0 = Polyline(pts)
            l1 = Bezier(pts, weights=wts)
        make_face()
    extrude(amount=10)
```

- **Algebra mode**

```
pts = [
    (0, 0),
    (20, 20),
    (40, 0),
    (0, -40),
    (-60, 0),
    (0, 100),
    (100, 0),
]

wts = [
    1.0,
    1.0,
    2.0,
    3.0,
    4.0,
    2.0,
    1.0,
]

ex30_ln = Polyline(pts) + Bezier(pts, weights=wts)
ex30_sk = make_face(ex30_ln)
ex30 = extrude(ex30_sk, -10)
```

1.8.31 31. Nesting Locations

Locations contexts can be nested to create groups of shapes. Here 24 triangles, 6 squares, and 1 hexagon are created and then extruded. Notably *PolarLocations* rotates any "children" groups by default.

- **Builder mode**

```
a, b, c = 80.0, 5.0, 3.0

with BuildPart() as ex31:
    with BuildSketch() as ex31_sk:
        with PolarLocations(a / 2, 6):
            with GridLocations(3 * b, 3 * b, 2, 2):
                RegularPolygon(b, 3)
            RegularPolygon(b, 4)
        RegularPolygon(3 * b, 6, rotation=30)
    extrude(amount=c)
```

- **Algebra mode**

```
a, b, c = 80.0, 5.0, 3.0

ex31 = Rot(Z=30) * RegularPolygon(3 * b, 6)
ex31 += PolarLocations(a / 2, 6) * (
    RegularPolygon(b, 4) + GridLocations(3 * b, 3 * b, 2, 2) *␣
↪RegularPolygon(b, 3)
)
ex31 = extrude(ex31, 3)
```

1.8.32 32. Python For-Loop

In this example, a standard python for-loop is used along with a list of faces extracted from a sketch to progressively modify the extrusion amount. There are 7 faces in the sketch, so this results in 7 separate calls to *extrude()*.

- **Builder mode**

  Mode .PRIVATE is used in *BuildSketch* to avoid adding these faces until the for-loop.

```
a, b, c = 80.0, 10.0, 1.0

with BuildPart() as ex32:
    with BuildSketch(mode=Mode.PRIVATE) as ex32_sk:
        RegularPolygon(2 * b, 6, rotation=30)
        with PolarLocations(a / 2, 6):
            RegularPolygon(b, 4)
    for idx, obj in enumerate(ex32_sk.sketch.faces()):
        add(obj)
        extrude(amount=c + 3 * idx)
```

- **Algebra mode**

```
a, b, c = 80.0, 10.0, 1.0

ex32_sk = RegularPolygon(2 * b, 6, rotation=30)
ex32_sk += PolarLocations(a / 2, 6) * RegularPolygon(b, 4)
ex32 = Part() + [extrude(obj, c + 3 * idx) for idx, obj in enumerate(ex32_
↪sk.faces())]
```

1.8.33 33. Python Function and For-Loop

Building on the previous example, a standard python function is used to return a sketch as a function of several inputs to progressively modify the size of each square.

- **Builder mode**

  The function returns a *BuildSketch*.

```
a, b, c = 80.0, 5.0, 1.0


def square(rad, loc):
    with BuildSketch() as sk:
        with Locations(loc):
            RegularPolygon(rad, 4)
    return sk.sketch


with BuildPart() as ex33:
    with BuildSketch(mode=Mode.PRIVATE) as ex33_sk:
        locs = PolarLocations(a / 2, 6)
        for i, j in enumerate(locs):
            add(square(b + 2 * i, j))
    for idx, obj in enumerate(ex33_sk.sketch.faces()):
        add(obj)
        extrude(amount=c + 2 * idx)
```

- **Algebra mode**

  The function returns a `Sketch` object.

```
a, b, c = 80.0, 5.0, 1.0


def square(rad, loc):
    return loc * RegularPolygon(rad, 4)


ex33 = Part() + [
    extrude(square(b + 2 * i, loc), c + 2 * i)
    for i, loc in enumerate(PolarLocations(a / 2, 6))
]
```

1.8.34 34. Embossed and Debossed Text

- **Builder mode**

  The text "Hello" is placed on top of a rectangle and embossed (raised) by placing a BuildSketch on the top face (`topf`). Note that *Align* is used to control the text placement. We re-use the `topf` variable to select the same face and deboss (indented) the text "World". Note that if we simply ran `BuildSketch(ex34.faces().sort_by(Axis.Z)[-1])` for both `ex34_sk1 & 2` it would incorrectly locate the 2nd "World" text on the top of the "Hello" text.

```
length, width, thickness, fontsz, fontht = 80.0, 60.0, 10.0, 25.0, 4.0

with BuildPart() as ex34:
    Box(length, width, thickness)
    topf = ex34.faces().sort_by(Axis.Z)[-1]
    with BuildSketch(topf) as ex34_sk:
        Text("Hello", font_size=fontsz, align=(Align.CENTER, Align.MIN))
    extrude(amount=fontht)
    with BuildSketch(topf) as ex34_sk2:
        Text("World", font_size=fontsz, align=(Align.CENTER, Align.MAX))
    extrude(amount=-fontht, mode=Mode.SUBTRACT)
```

- **Algebra mode**

  The text "Hello" is placed on top of a rectangle and embossed (raised) by placing a sketch on the top face (`topf`). Note that *Align* is used to control the text placement. We re-use the `topf` variable to select the same face and deboss (indented) the text "World".

```
length, width, thickness, fontsz, fontht = 80.0, 60.0, 10.0, 25.0, 4.0

ex34 = Box(length, width, thickness)
plane = Plane(ex34.faces().sort_by().last)
ex34_sk = plane * Text("Hello", font_size=fontsz, align=(Align.CENTER,␣
↪Align.MIN))
ex34 += extrude(ex34_sk, amount=fontht)
ex34_sk2 = plane * Text("World", font_size=fontsz, align=(Align.CENTER,␣
↪Align.MAX))
ex34 -= extrude(ex34_sk2, amount=-fontht)
```

1.8.35 35. Slots

- **Builder mode**

  Here we create a *SlotCenterToCenter* and then use a *BuildLine* and *RadiusArc* to create an arc for two instances of *SlotArc*.

```
length, width, thickness = 80.0, 60.0, 10.0

with BuildPart() as ex35:
    Box(length, length, thickness)
    topf = ex35.faces().sort_by(Axis.Z)[-1]
    with BuildSketch(topf) as ex35_sk:
        SlotCenterToCenter(width / 2, 10)
        with BuildLine(mode=Mode.PRIVATE) as ex35_ln:
            RadiusArc((-width / 2, 0), (0, width / 2), radius=width / 2)
        SlotArc(arc=ex35_ln.edges()[0], height=thickness, rotation=0)
        with BuildLine(mode=Mode.PRIVATE) as ex35_ln2:
            RadiusArc((0, -width / 2), (width / 2, 0), radius=-width / 2)
        SlotArc(arc=ex35_ln2.edges()[0], height=thickness, rotation=0)
    extrude(amount=-thickness, mode=Mode.SUBTRACT)
```

- **Algebra mode**

Here we create a *SlotCenterToCenter* and then use a *RadiusArc* to create an arc for two instances of SlotArc.

```
length, width, thickness = 80.0, 60.0, 10.0

ex35 = Box(length, length, thickness)
plane = Plane(ex35.faces().sort_by().last)
ex35_sk = SlotCenterToCenter(width / 2, 10)
ex35_ln = RadiusArc((-width / 2, 0), (0, width / 2), radius=width / 2)
ex35_sk += SlotArc(arc=ex35_ln.edges()[0], height=thickness)
ex35_ln2 = RadiusArc((0, -width / 2), (width / 2, 0), radius=-width / 2)
ex35_sk += SlotArc(arc=ex35_ln2.edges()[0], height=thickness)
ex35 -= extrude(plane * ex35_sk, -thickness)
```

1.8.36 36. Extrude Until

Sometimes you will want to extrude until a given face that could be non planar or where you might not know easily the distance you have to extrude to. In such cases you can use *extrude() Until* with Until.NEXT or Until.LAST.

- **Builder mode**

```
rad, rev = 6, 50

with BuildPart() as ex36:
    with BuildSketch() as ex36_sk:
        with Locations((0, rev)):
            Circle(rad)
    revolve(axis=Axis.X, revolution_arc=180)
    with BuildSketch() as ex36_sk2:
        Rectangle(rad, rev)
    extrude(until=Until.NEXT)
```

- **Algebra mode**

```
rad, rev = 6, 50

ex36_sk = Pos(0, rev) * Circle(rad)
ex36 = revolve(axis=Axis.X, profiles=ex36_sk, revolution_arc=180)
ex36_sk2 = Rectangle(rad, rev)
ex36 += extrude(ex36_sk2, until=Until.NEXT, target=ex36)
```

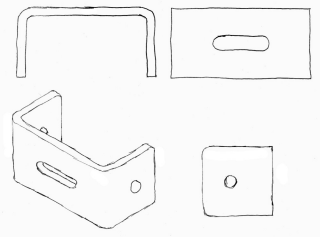
1.9 Tutorials

There are several tutorials to help guide uses through the concepts of build123d in a step by step way. Working through these tutorials in order is recommended as later tutorials build on the concepts introduced in earlier ones.

1.9.1 Designing a Part in build123d

Designing a part with build123d involves a systematic approach that leverages the power of 2D profiles, extrusions, and revolutions. Where possible, always work in the lowest possible dimension, 1D lines before 2D sketches before 3D parts. The following guide will get you started:

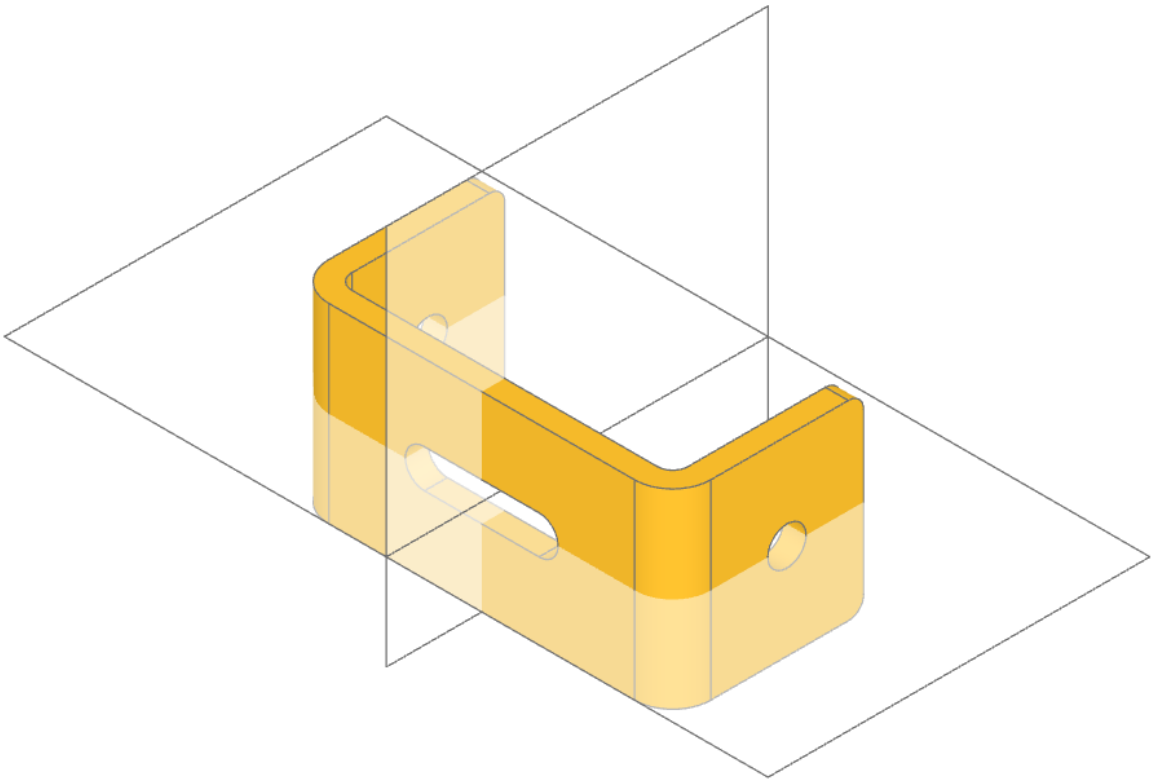
As an example, we'll go through the design process for this bracket:



Step 1. Examine the Part in All Three Orientations

Start by visualizing the part from the front, top, and side views. Identify any symmetries in these orientations, as symmetries can simplify the design by reducing the number of unique features you need to model.

In the following view of the bracket one can see two planes of symmetry so we'll only need to design one quarter of it.



Step 2. Identify Rotational Symmetries

Look for structures that could be created through the rotation of a 2D shape. For instance, cylindrical or spherical features are often the result of revolving a profile around an axis. Identify the axis of rotation and make a note of it.

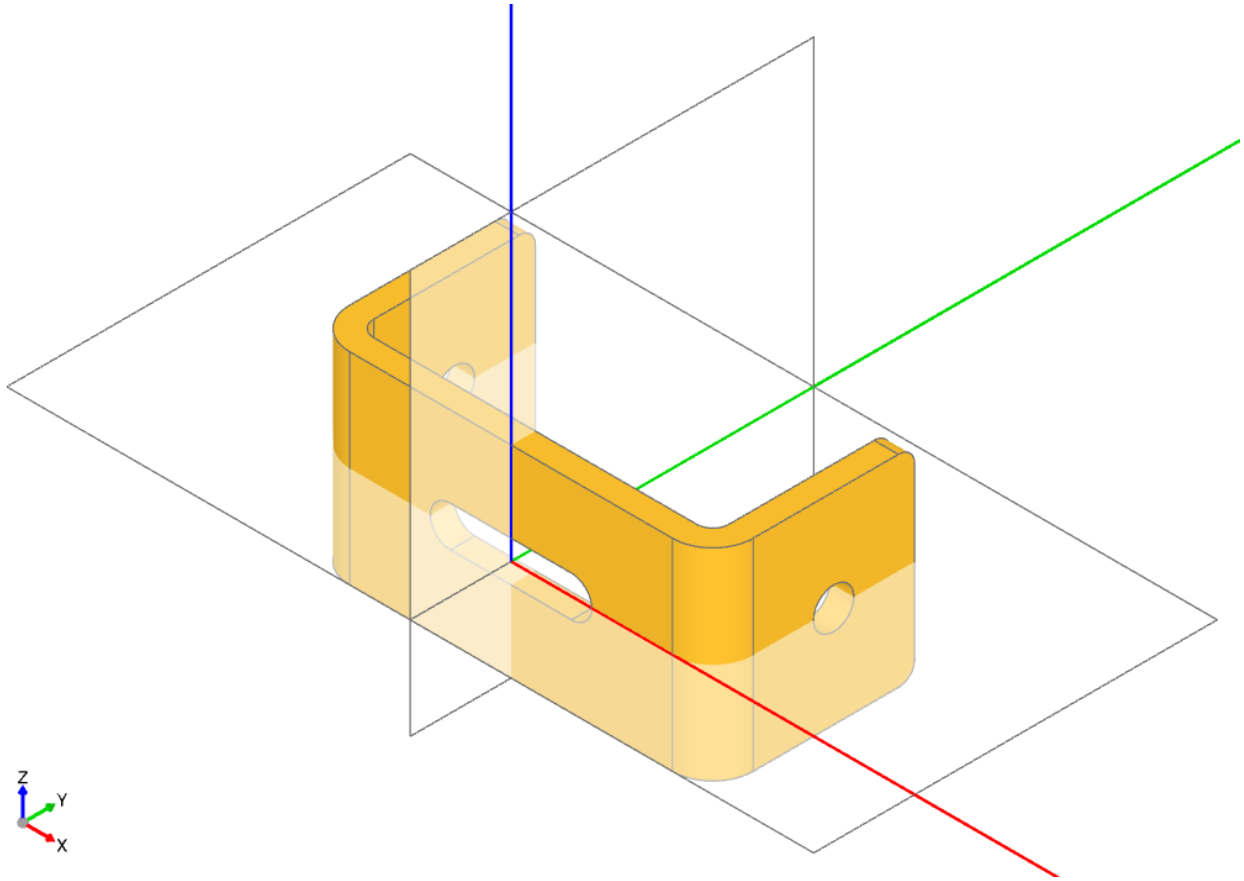
There are no rotational structures in the example bracket.

Step 3. Select a Convenient Origin

Choose an origin point that minimizes the need to move or transform components later in the design process. Ideally, the origin should be placed at a natural center of symmetry or a critical reference point on the part.

The planes of symmetry for the bracket was identified in step 1, making it logical to place the origin at the intersection of these planes on the bracket's front face. Additionally, we'll define the coordinate system we'll be working in: Plane.XY

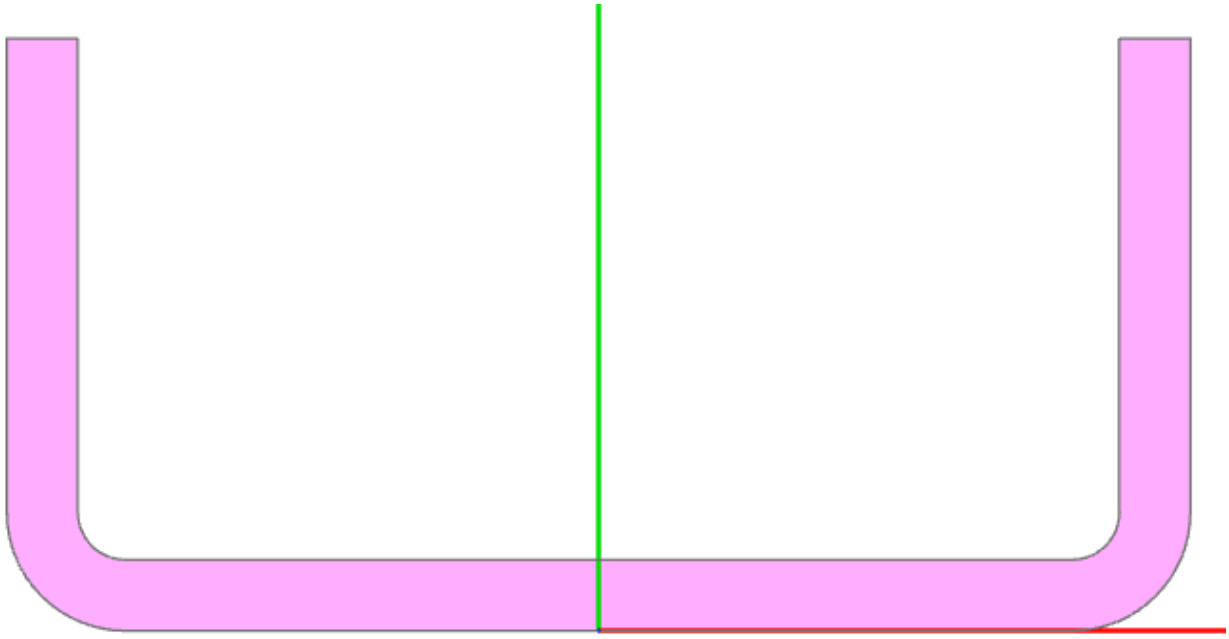
(the default), where the origin is set at the global (0,0,0) position. In this system, the x-axis aligns with the front of the bracket, and the z-axis corresponds to its width. It's important to note that all coordinate systems/planes in build123d adhere to the right-hand rule *meaning the y-axis is automatically determined by this convention.*



Step 4. Create 2D Profiles

Design the 2D profiles of your part in the appropriate orientation(s). These profiles are the foundation of the part's geometry and can often represent cross-sections of the part. Mirror parts of profiles across any axes of symmetry identified earlier.

The 2D profile of the bracket is as follows:



The build123d code to generate this profile is as follows:

```
with BuildSketch() as sketch:
    with BuildLine() as profile:
        FilletPolyline(
            (0, 0), (length / 2, 0), (length / 2, height), radius=bend_radius
        )
        offset(amount=thickness, side=Side.LEFT)
    make_face()
    mirror(about=Plane.YZ)
```

This code creates a 2D sketch of a mirrored profile in the build123d CAD system. Here's a step-by-step explanation of what it does:

with BuildSketch() as sketch:
This starts a context for creating a 2D sketch, which defines the overall boundary and geometric features. The sketch will be stored in the variable sketch.

with BuildLine() as profile:
This starts another context, this time for drawing lines (or profiles) within the sketch. The profile consists of connected line segments, arcs, or polylines.

FilletPolyline((0, 0), (length / 2, 0), (length / 2, height), radius=bend_radius)
This object draws a polyline with three points: (0,0), (length/2, 0), and (length/2, height). A fillet (curved corner) with a radius of bend_radius is added where applicable between the segments of the polyline.

offset(amount=thickness, side=Side.LEFT)
This applies an offset to the polyline created earlier. The offset creates a parallel line at a distance of thickness to the left side of the original polyline. This operation essentially thickens the profile by a given amount.

make_face()
This command creates a 2D face from the closed profile. The offset operation ensures that the profile is closed, allowing the creation of a solid face from the boundary defined.

mirror(about=Plane.YZ)
This mirrors the entire face about the YZ plane (which runs along the center of the sketch), creating a symmetrical counterpart of the face. The mirrored geometry will complete the final shape.

Step 5. Use Extrusion for Prismatic Features

For solid or prismatic shapes, extrude the 2D profiles along the necessary axis. You can also combine multiple extrusions by intersecting or unionizing them to form complex shapes. Use the resulting geometry as sub-parts if needed.

The next step in implementing our design in build123d is to convert the above sketch into a part by extruding it as shown in this code:

```
with BuildPart() as bracket:
    with BuildSketch() as sketch:
        with BuildLine() as profile:
            FilletPolyline(
                (0, 0), (length / 2, 0), (length / 2, height), radius=bend_radius
            )
            offset(amount=thickness, side=Side.LEFT)
        make_face()
        mirror(about=Plane.YZ)
    extrude(amount=width / 2)
    mirror(about=Plane.XY)
```

In this example, we've wrapped the sketch within a BuildPart context, which is used for creating 3D parts. We utilized the extrude function to extend the 2D sketch into a solid object, turning it into a 3D part. Additionally, we applied the mirror function to replicate the partial part across a plane of symmetry, ensuring a symmetrical design.

Step 6. Generate Revolved Features

If any part of the geometry can be created by revolving a 2D profile around an axis, use the revolve operation. This is particularly useful for parts that include cylindrical, conical, or spherical features. Combine these revolved sub-parts with existing features using additive, subtractive, or intersecting operations.

Our example has no revolved features.

Step 7. Combine Sub-parts Intelligently

When combining multiple sub-parts, keep in mind whether they need to be added, subtracted, or intersected. Subtracting or intersecting can create more refined details, while addition is useful for creating complex assemblies.

Out example only has one sub-part but further sub-parts could be created in the BuildPart context by defining more sketches and extruding or revolving them.

Step 8. Apply Chamfers and Fillets

Identify critical edges or vertices that need chamfering or filleting. Use build123d's selectors to apply these operations accurately. Always visually inspect the results to ensure the correct edges have been modified.

The back corners of the bracket need to be rounded off or filleted so the edges that define these corners need to be isolated. The following code, placed to follow the previous code block, captures just these edges:

```
corners = bracket.edges().filter_by(Axis.X).group_by(Axis.Y)[-1]
fillet(corners, fillet_radius)
```

These lines isolates specific corner edges that are then filleted.

corners = bracket.edges().filter_by(Axis.X).group_by(Axis.Y)[-1]
This line is used to select specific edges from the 3D part (bracket) that was created by the extrusion.

- bracket.edges() *retrieves all the edges of the bracket part.*
- filter_by(Axis.X) *filters the edges to only those that are aligned along the X-axis.*
- group_by(Axis.Y) *groups the edges by their positions along the Y-axis. This operation essentially organizes the filtered X-axis edges into groups based on their Y-coordinate positions.*
- [-1] *selects the last group of edges along the Y-axis, which corresponds to the back of the part - the edges we are looking for.*

fillet(corners, fillet_radius)
This function applies a fillet (a rounded edge) to the selected corners, with a specified radius (fillet_radius). The fillet smooths the sharp edges at the corners, giving the part a more refined shape.

Step 9. Design for Assembly

If the part is intended to connect with others, add features like joints, holes, or other attachment points. Ensure that these features are precisely located to ensure proper fitment and functionality in the final assembly.

Our example has two circular holes and a slot that need to be created. First we'll create the two circular holes:

```
with Locations(bracket.faces().sort_by(Axis.X)[-1]):
    Hole(hole_diameter / 2)
```

This code creates a hole in a specific face of the bracket part.

with Locations(bracket.faces().sort_by(Axis.X)[-1]):
This context sets a location(s) for subsequent operations.

- bracket.faces() *retrieves all the faces of the bracket part.*
- sort_by(Axis.X) *sorts these faces based on their position along the X-axis (from one side of the bracket to the other).*
- [-1] *selects the last face in this sorted list, which would be the face farthest along the X-axis, the extreme right side of the part.*
- Locations() *creates a new local context or coordinate system at the selected face, effectively setting this face as the working location for any subsequent operations inside the with block.*

Hole(hole_diameter / 2)
This creates a hole in the selected face. The radius of the hole is specified as hole_diameter / 2. The hole is placed at the origin of the selected face, based on the local coordinate system created by Locations(). As the depth of the hole is not provided it is assumed to go entirely through the part.

Next the slot needs to be created in the bracket with will be done by sketching a slot on the front of the bracket and extruding the sketch through the part.

```
with BuildSketch(bracket.faces().sort_by(Axis.Y)[0]):
    SlotOverall(20 * MM, hole_diameter)
extrude(amount=-thickness, mode=Mode.SUBTRACT)
```

Here's a detailed explanation of what each part does:

with BuildSketch(bracket.faces().sort_by(Axis.Y)[0]):
This line sets up a sketching context.

- bracket.faces() *retrieves all the faces of the bracket part.*

- sort_by(Axis.Y) *sorts the faces along the Y-axis, arranging them from the lowest Y-coordinate to the highest.*
- [0] *selects the first face in this sorted list, which is the one located at the lowest Y-coordinate, the nearest face of the part.*
- BuildSketch() *creates a new sketching context on this selected face, where 2D geometry will be drawn.*

SlotOverall(20, hole_diameter)
This command draws a slot (a rounded rectangle or elongated hole) on the selected face. The slot has a total length of 20 mm and a width equal to hole_diameter. The slot is defined within the 2D sketch on the selected face of the bracket.

extrude(amount=-thickness, mode=Mode.SUBTRACT)
extrude() *takes the 2D sketch (the slot) and extends it into the 3D space by a distance equal to -thickness, creating a cut into the part. The negative value (-thickness) indicates that the extrusion is directed inward into the part (a cut).* mode=Mode.SUBTRACT *specifies that the extrusion is a subtractive operation, meaning it removes material from the bracket, effectively cutting the slot through the face of the part.*

Although beyond the scope of this tutorial, joints could be defined for each of the holes to allow programmatic connection to other parts.

Step 10. Plan for Parametric Flexibility

Wherever possible, make your design parametric, allowing dimensions and features to be easily adjusted later. This flexibility can be crucial if the design needs modifications or if variations of the part are needed.

The dimensions of the bracket are defined as follows:

```
thickness = 3 * MM
width = 25 * MM
length = 50 * MM
height = 25 * MM
hole_diameter = 5 * MM
bend_radius = 5 * MM
fillet_radius = 2 * MM
```

Step 11. Test Fit and Tolerances

Visualize the fit of the part within its intended assembly. Consider tolerances for manufacturing, such as clearance between moving parts or shrinkage for 3D-printed parts. Adjust the design as needed to ensure real-world functionality.

Summary

These steps should guide you through a logical and efficient workflow in build123d (or any CAD tool), helping you to design parts with accuracy and ease.

The entire code block for the bracket example is shown here:

```
from build123d import *
from ocp_vscode import show_all

thickness = 3 * MM
width = 25 * MM
length = 50 * MM
```

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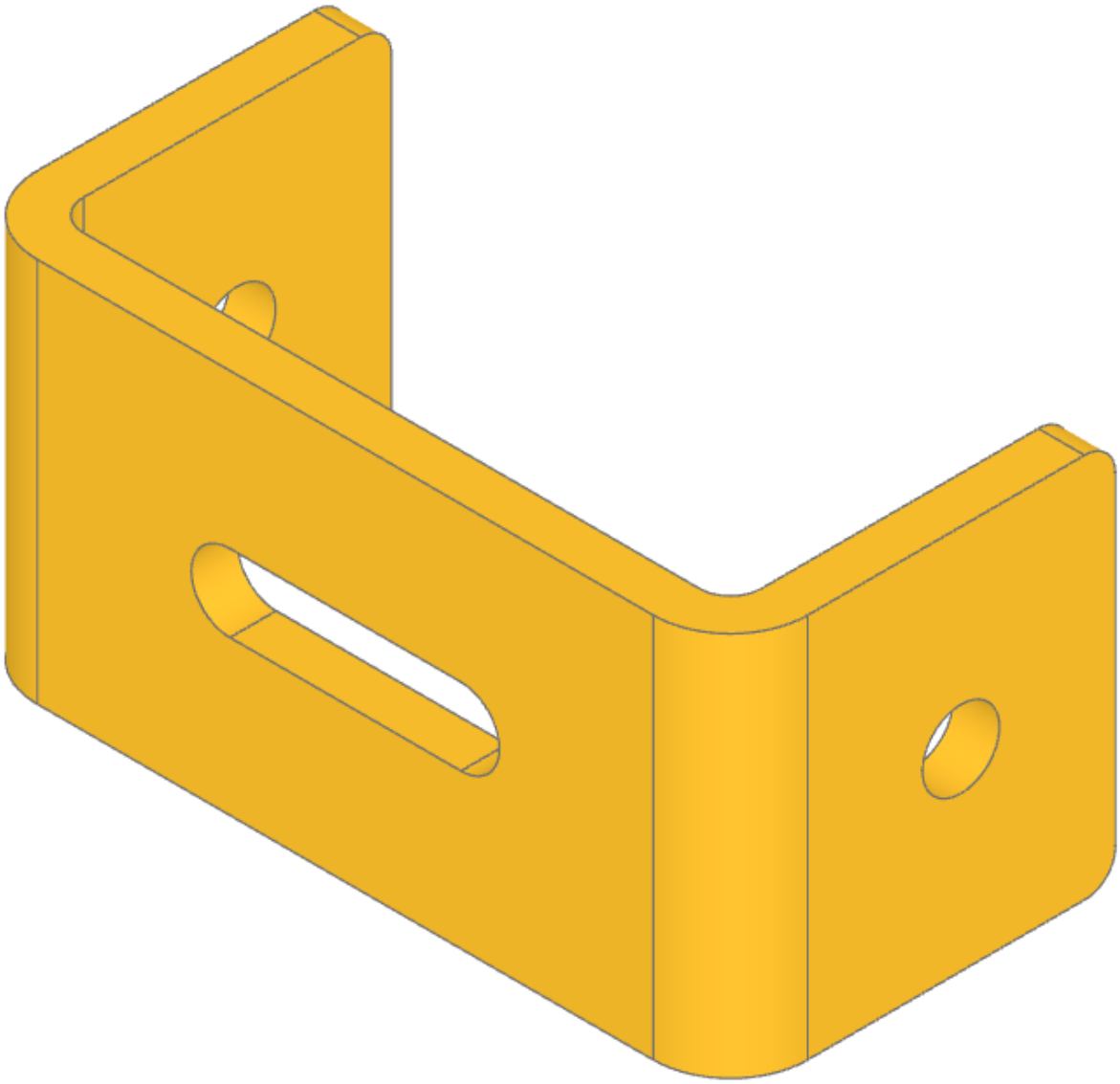
```
height = 25 * MM
hole_diameter = 5 * MM
bend_radius = 5 * MM
fillet_radius = 2 * MM

with BuildPart() as bracket:
    with BuildSketch() as sketch:
        with BuildLine() as profile:
            FilletPolyline(
                (0, 0), (length / 2, 0), (length / 2, height), radius=bend_radius
            )
            offset(amount=thickness, side=Side.LEFT)
        make_face()
        mirror(about=Plane.YZ)
    extrude(amount=width / 2)
    mirror(about=Plane.XY)
    corners = bracket.edges().filter_by(Axis.X).group_by(Axis.Y)[-1]
    fillet(corners, fillet_radius)
    with Locations(bracket.faces().sort_by(Axis.X)[-1]):
        Hole(hole_diameter / 2)
    with BuildSketch(bracket.faces().sort_by(Axis.Y)[0]):
        SlotOverall(20 * MM, hole_diameter)
    extrude(amount=-thickness, mode=Mode.SUBTRACT)

show_all()
```



1.9.2 Selector Tutorial

This tutorial provides a step by step guide in using selectors as we create this part:

> **Note**
>
> One can see any object in the following tutorial by using the `ocp_vscode` (or any other supported viewer) by using the `show(object_to_be_viewed)` command. Alternatively, the `show_all()` command will display all objects that have been assigned an identifier.

Step 1: Setup

Before getting to the CAD operations, this selector script needs to import the build123d environment.

```
from build123d import *
from ocp_vscode import *
```

Step 2: Create Base with BuildPart

To start off, the part will be based on a cylinder so we'll use the *Cylinder* object of *BuildPart*:

```
from build123d import *
from ocp_vscode import *

with BuildPart() as example:
    Cylinder(radius=10, height=3)
```

Step 3: Place Sketch on top of base

The next set of features in this design will be created on the top of the cylinder and be described by a planar sketch (*BuildSketch* is the tool for drawing on planar surfaces) , so we'll create a sketch centered on the top of the cylinder. To locate this sketch we'll use the cylinder's top Face as shown here:

```
from build123d import *
from ocp_vscode import *

with BuildPart() as example:
    Cylinder(radius=10, height=3)
    with BuildSketch(example.faces().sort_by(Axis.Z)[-1]):
```

Here we're using selectors to find that top Face - let's break down `example.faces().sort_by(Axis.Z)[-1]`:

Step 3a: Extract Faces from a part

The first sub-step is the extraction of all of the Faces from the part that we're building. The *BuildPart* instance was assigned the identifier `example` so `example.faces()` will extract all of the Faces from that part into a custom python `list` - a *ShapeList*.

Step 3b: Get top Face

The next sub-step is to sort the ShapeList of Faces by their position with respect to the Z Axis. The `sort_by` method will sort the list by relative position of the object's center to the `Axis.Z` and `[-1]` selects the last item on that list - or return the top Face of the `example` part.

Step 4: Create hole shape

The object has a hexagonal hole in the top with a central cylinder which we'll describe in the sketch.

```
from build123d import *
from ocp_vscode import *

with BuildPart() as example:
    Cylinder(radius=10, height=3)
    with BuildSketch(example.faces().sort_by(Axis.Z)[-1]):
```

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```
        RegularPolygon(radius=7, side_count=6)
        Circle(radius=4, mode=Mode.SUBTRACT)
```

Step 4a: Draw a hexagon

We'll create a hexagon with the use of *RegularPolygon* object with six sides.

Step 4b: Create a hole in the hexagon

To create the hole we'll subtract a *Circle* from the sketch by using `mode=Mode.SUBTRACT`. The sketch now described the hexagonal hole that we want to make in the *Cylinder*.

Step 5: Create the hole

To create the hole we'll *extrude()* the sketch we just created into the *Cylinder* and subtract it.

```
from build123d import *
from ocp_vscode import *

with BuildPart() as example:
    Cylinder(radius=10, height=3)
    with BuildSketch(example.faces().sort_by(Axis.Z)[-1]):
        RegularPolygon(radius=7, side_count=6)
        Circle(radius=4, mode=Mode.SUBTRACT)
    extrude(amount=-2, mode=Mode.SUBTRACT)
```

Note that `amount=-2` indicates extruding into the part and - just like with the sketch - `mode=Mode.SUBTRACT` instructs the builder to subtract this hexagonal shape from the part under construction.

At this point the part looks like:

Step 6: Fillet the top perimeter Edge

The final step is to apply a fillet to the top perimeter.

```
from build123d import *
from ocp_vscode import *

with BuildPart() as example:
    Cylinder(radius=10, height=3)
    with BuildSketch(example.faces().sort_by(Axis.Z)[-1]):
        RegularPolygon(radius=7, side_count=6)
        Circle(radius=4, mode=Mode.SUBTRACT)
    extrude(amount=-2, mode=Mode.SUBTRACT)
    fillet(
        example.edges()
        .filter_by(GeomType.CIRCLE)
        .sort_by(SortBy.RADIUS)[-2:]
        .sort_by(Axis.Z)[-1],
        radius=1,
    )
```

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```
show(example)
```

Here we're using the *fillet()* operation which needs two things: the edge(s) to fillet and the radius of the fillet. To provide the edge, we'll use more selectors as described in the following sub-steps.

Step 6a: Extract all the Edges

Much like selecting Faces in Step 3a, we'll select all of the `example` part's edges with `example.edges()`.

Step 6b: Filter the Edges for circles

Since we know that the edge we're looking for is a circle, we can filter the edges selected in Step 6a for just those that are of geometric type `CIRCLE` with `example.edges().filter_by(GeomType.CIRCLE)`. This step removes all of the Edges of the hexagon hole.

Step 6c: Sort the circles by radius

The perimeter are the largest circles - the central cylinder must be excluded - so we'll sort all of the circles by their radius with: `example.edges().filter_by(GeomType.CIRCLE).sort_by(SortBy.RADIUS)`.

Step 6d: Slice the list to get the two largest

We know that the `example` part has two perimeter circles so we'll select just the top two edges from the sorted circle list with: `example.edges().filter_by(GeomType.CIRCLE).sort_by(SortBy.RADIUS)[-2:]`. The syntax of this slicing operation is standard python list slicing.

Step 6e: Select the top Edge

The last sub-step is to select the top perimeter edge, the one with the greatest Z value which we'll do with the `sort_by(Axis.Z)[-1]` method just like Step 3b - note that these methods work on all Shape objects (Edges, Wires, Faces, Solids, and Compounds) - with: `example.edges().filter_by(GeomType.CIRCLE).sort_by(SortBy.RADIUS)[-2:].sort_by(Axis.Z)[-1]`.

Conclusion

By using selectors as we have in this example we've used methods of identifying features that are robust to features changing within the part. We've also avoided the classic CAD "Topological naming problem" by never referring to features with names or tags that could become obsolete as the part changes.

When possible, avoid using static list indices to refer to features extracted from methods like `edges()` as the order within the list is not guaranteed to remain the same.

1.9.3 Drawing with Constraints

Introduction

CAD constraints are geometric and dimensional rules that define how sketch or assembly entities relate to one another. They control degrees of freedom (for example, parallel, perpendicular, tangent, coincident, distance, or angle), so edits preserve design intent instead of introducing unintended shape changes. This is the foundation of parametric modeling: behavior is driven by explicit relationships, not fixed manually drawn geometry. This section only addresses sketch constraints.

In graphical CAD systems, sketching is usually a two-step workflow: first draw approximate geometry, then add dimensions and constraints so a global solver can infer exact positions. That model works well for interactive drawing,

but it also encourages tightly coupled constraint networks that can become difficult to predict and maintain as a design evolves. It is also typically strongest for lines and circular arcs, with more limited and less robust behavior for ellipses, splines, and other higher-order curves.

In build123d, the primary workflow is different. Geometry is defined precisely at creation time using coordinates, parameters, and explicit geometric relationships in code. Instead of building a large interdependent constraint graph and asking a global solver to resolve it, you express intent directly: mirror about a plane, construct tangent features, derive points and frames from existing topology, and compose operations deterministically.

This does not eliminate constrained construction; it scopes it. build123d provides targeted geometric local solvers for common high-value problems, including objects such as *BlendCurve*, *ConstrainedLines*, *ConstrainedArcs*, and *Triangle*. It also provides a growing family of constructors whose extent can be determined by other geometry, including *PolarLine*, *CenterArc*, *EllipticalCenterArc*, *EllipticalStartArc*, *JernArc*, *ParabolicCenterArc*, and *HyperbolicCenterArc*. Together with operations such as *make_hull*, *mirror*, and *offset*, these tools solve specific constraint patterns while keeping model behavior explicit, deterministic, and readable in code.

The result is a practical hybrid approach: precise programmatic modeling by default, with specialized constrained constructors when they provide clear leverage. For most production parts, this yields robust, maintainable sketches without the overhead and fragility of a general-purpose sketch solver.

Constraint Types

build123d supports several practical forms of constrained construction. Rather than relying on a single global sketch solver, it provides targeted tools that enforce specific geometric relationships directly and predictably.

Analytical Constraints

Triangle
: Constructs a triangle from any three parameters (side lengths and/or interior angles) and solves for the others. Angle naming follows standard convention: side `a` is opposite angle `A`, side `b` is opposite angle `B`, and side `c` is opposite angle `C`.

Continuity Constraints

BlendCurve
: Creates a smooth Bézier transition between two existing edges.

  In this context, *continuity* describes how smoothly the new blend joins the input edges at each endpoint:

  - C0 (positional continuity): endpoints meet, but direction may kink.
  - C1 (tangent continuity): endpoints and tangent directions match, giving a visually smooth join with no corner.
  - C2 (curvature continuity): endpoints, tangents, and curvature trend match, reducing curvature jumps and producing a smoother fairing.

  BlendCurve builds a Bézier curve that satisfies these endpoint constraints:

  - cubic Bézier for C1 blending (position + first derivative),
  - quintic Bézier for C2 blending (position + first and second derivatives).

  The derivatives are sampled from the two source edges at the selected connection points, then converted into Bézier control points that enforce the requested continuity. Optional tangent scaling factors let you tune how strongly the blend departs from each source edge, which adjusts perceived tension and transition shape without changing the endpoint constraints.

Geometric Relationship Constraints

@ and % operators
: Use @ (position-at) and % (tangent-at) to construct geometry relative to existing geometry. Typical uses include starting a new edge at an exact point on another edge, or aligning a new edge direction to a sampled tangent.

mirror
: Enforces symmetry by reflecting geometry about a plane, producing mirrored entities with exact geometric correspondence to the source.

Extent / Termination Constraints

PolarLine, *CenterArc*, *EllipticalCenterArc*, *EllipticalStartArc*, *JernArc*, *ParabolicCenterArc*, **and** *HyperbolicCenterArc*
: Construct curves from natural geometric parameters, then let another object determine where the result ends.

  In these constructors, the size argument can often be either:

  - a numeric angular or linear extent, or
  - a limiting object such as a *Shape*, *Axis*, *Location*, *Plane*, or point-like object.

  When a limit object is provided, the constructor creates the candidate geometry from the supplied start conditions, trims it at the first valid intersection with the limit, and returns the shortest valid result from the start. If no valid intersection exists, a `ValueError` is raised.

  This pattern is especially useful when design intent is "go in this direction until you meet that object", because it removes helper construction lines and separate trim calls while keeping the relationship local to the constructor call.

Offset / Equidistance Constraints

offset
: Creates geometry at a constant normal distance from a source edge or wire.

  This enforces an equidistance relationship commonly used for wall thickness, clearances, toolpaths, and parallel profile construction. Join behavior (for example at corners) can be controlled to match the design intent.

Tangency Constraints

ConstrainedArcs **and** *ConstrainedLines*
: Provide local constrained solving for 2D line-and-circle constructions. These APIs solve common geometric construction problems from explicit numeric and geometric constraints relative to existing curves.

  Supported constraint patterns include:

  - circle with specified radius,
  - line at a specified angle to another line,
  - tangency of a line or circle to a reference curve,
  - line or circle passing through a point,
  - circle center constrained to a point or to lie on a curve.

  For example, you can construct a circle with a given radius whose center lies on a specified line and which is tangent to another circle. This style of targeted solving covers high-value sketch workflows while keeping branch selection explicit and deterministic in code.

Multiple Solutions and Qualification

Tangency construction is typically multi-solution. A single problem statement can produce several valid geometric branches depending on where the solution lies relative to the reference entities.

For example, a circle of fixed radius tangent to two secant circles can produce up to eight valid solutions as shown below. This is expected behavior, not an error.

To reduce ambiguity, tangency constraints support **qualification** of relative position:

- `Tangency.ENCLOSING`: the solution must enclose the argument.
- `Tangency.ENCLOSED`: the solution must be enclosed by the argument.
- `Tangency.OUTSIDE`: the solution and argument must be external to each other.
- `Tangency.UNQUALIFIED`: no positional filtering; all valid branches are returned.

These qualifiers are intuitive for circles (inside/outside). For general oriented curves, interior is defined as the left-hand side of the curve with respect to its orientation.

Even with qualification, more than one solution may remain. In that case, use a `selector` to choose deterministic outputs.

Selecting results

In Algebra mode, select from returned edges after construction:

```
arcs = ConstrainedArcs(..., sagitta=Sagitta.BOTH)
chosen = arcs.edges().sort_by(Edge.length)[0]
```

In Builder mode, prefer the constructor `selector` argument so only desired branches are added to the active context:

```
with BuildLine():
   ConstrainedArcs(
         ...,
         selector=lambda edges: edges.sort_by_distance((0, 0))[0],
   )
```

This combination of qualification + selection gives robust, explicit control over tangency branch choice.

Practical Examples

The following examples show how each constraint type is used in production-style sketching. Each example is intentionally small, with construction geometry kept visible in code so the relationship logic is explicit and reusable.

Analytical Constraints

build123d includes a built-in *Triangle* object that has an internal solver such that one can specify any three parameters of a triangle and solve for the others. For example:

```
>>> isosceles = Triangle(a=30, b=30, C=60)
>>> isosceles.c
29.999999999999996
>>> isosceles.A
60.00000000000001
>>> isosceles.B
```

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```
60.00000000000001
>>> isosceles.vertex_A
Vertex(-1.7763568394002505e-15, 17.32050807568877, 0.0)
```

In this example, side lengths `a` and `b` with included angle `C` are provided. The object then computes the remaining side, angles, and vertices. This is useful when a design intent is naturally expressed as triangle dimensions instead of explicit coordinates.

One can easily use external solvers, say the symbolic solver `sympy`, within your build123d code as follows:

```
from math import sin, cos, tan, radians
from build123d import *
from ocp_vscode import *
import sympy

# This problem uses the sympy symbolic math solver

# Define the symbols for the unknowns
# - the center of the radius 30 arc (x30, y30)
# - the center of the radius 66 arc (x66, y66)
# - end of the 8° line (l8x, l8y)
# - the point with the radius 30 and 66 arc meet i30_66
# - the start of the horizontal line lh
y30, x66, xl8, yl8 = sympy.symbols("y30 x66 xl8 yl8")
x30 = 77 - 55 / 2
y66 = 66 + 32

# There are 4 unknowns so we need 4 equations
equations = [
   (x66 - x30) ** 2 + (y66 - y30) ** 2 - (66 + 30) ** 2,  # distance between centers
   xl8 - (x30 + 30 * sin(radians(8))),  # 8 degree slope
   yl8 - (y30 + 30 * cos(radians(8))),  # 8 degree slope
   (yl8 - 50) / (55 / 2 - xl8) - tan(radians(8)),  # 8 degree slope
]
# There are two solutions but we want the 2nd one
solution = {k: float(v) for k,v in sympy.solve(equations, dict=True)[1].items()}

# Create the critical points
c30 = Vector(x30, solution[y30])
c66 = Vector(solution[x66], y66)
l8 = Vector(solution[xl8], solution[yl8])

...
```

This pattern is useful when the governing relationships are algebraic but awkward to construct directly with primitives. Solve unknown parameters first, then feed the solved values into standard build123d geometry construction.

Continuity Constraints

One may want to join two curves with a third curve such that the connector satisfies a given continuity where they meet as shown here where a semi-circle (on the left) is joined to a spline (on the right).

```
m1 = CenterArc((-2, 0.6), 1, -10, 200).reversed()
m2 = Spline((0.4, -0.6), (1, -1.6), (2, 0))
connector = BlendCurve(m1, m2, tangent_scalars=(2, 1), continuity=ContinuityLevel.C2)
comb = Curve(Wire([m1, connector, m2]).curvature_comb(200))
```

The key call is `BlendCurve(..., continuity=ContinuityLevel.C2)`. `C2` continuity matches endpoint curvature trend in addition to position and tangent, which reduces visible fairness breaks at joins. `tangent_scalars` controls how strongly the connector departs from each source curve.

`curvature_comb` is used here as a diagnostic. The normal "comb" lines represent local curvature magnitude; smoother transitions produce gradual comb variation rather than abrupt spikes.

Geometric Relationship Constraints

Coincident

```
with BuildLine() as coincident_ex:
    l1 = Line((0, 0), (1, 2))
    l2 = Line(l1 @ 1, l1 @ 1 + (1, 0))
```

The second line starts at `l1 @ 1` (the end of `l1`), creating an exact coincident relationship without a separate constraint object.

Tangent

```
with BuildLine() as tangent_ex:
    l1 = Line((0, 0), (1, 1))
    l2 = JernArc(start=l1 @ 1, tangent=l1 % 1, radius=1, arc_size=70)
```

The arc starts at the line endpoint and uses `l1 % 1` as its initial tangent direction. This is a direct tangent construction: continuity is encoded in the creation call.

Perpendicular

```
with BuildLine() as perpendicular_ex:
    l1 = CenterArc((0, 0), 1.5, 0, 45)
    l2 = PolarLine(
        start=l1 @ 1, length=1, direction=l1.tangent_at(1).rotate(Axis.Z, -90)
    )
```

The direction vector is built from `l1.tangent_at(1)` rotated by 90 degrees, giving an explicit perpendicular relationship relative to curve orientation.

Extent / Termination Constraints

```
with BuildLine() as intersect_ex:
    c1 = EllipticalCenterArc((0, 0), 1.2, 1.8, 0, arc_size=120, mode=Mode.PRIVATE)
    l1 = PolarLine(start=(-0.2, 0.1), length=c1, angle=10)
```

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```
    l2 = PolarLine(start=(-0.2, 0.1), length=c1, angle=70)
    l3 = add(c1.trim(l1 @ 1, l2 @ 1))
```

PolarLine creates each line from a start point and direction, then limits it by intersection with the ellipse. This is often cleaner than creating long helper lines and manually trimming afterward, and the same pattern applies to a wide range of arcs and conics.

The same extent-by-object pattern works with several curved constructors:

- *CenterArc*
- *EllipticalCenterArc*
- *EllipticalStartArc*
- *JernArc*
- *ParabolicCenterArc*
- *HyperbolicCenterArc*

For example, a parabola or hyperbola can be grown from a start condition and terminated by a line or axis in the same way:

```
p1 = ParabolicCenterArc((0, 0), 0.5, 0, arc_size=Line((0, 1), (5, 1)))
h1 = HyperbolicCenterArc((0, 0), 2, 1, 0, arc_size=Axis((0, 1), (1, 0)))
```

This is particularly useful when sketches are not symmetric and multiple local constructions must terminate against different surrounding geometry.

Offset / Equidistance Constraints

```
inside = FilletPolyline((1.5, 0), (1.5, 1), (-1.5, 1), (-1.5, 0), radius=0.2)
perimeter = offset(inside, amount=0.2, side=Side.RIGHT)
```

offset preserves the source profile shape while enforcing constant wall thickness. This is a common pattern for clearances, shells, and manufacturing margins.

Tangency Constraints

Both *ConstrainedArcs* and *ConstrainedLines* return a *Curve* containing one or more *Edge* objects.

These constructors solve tangent/contact problems from mixed numeric and geometric inputs. Because tangency is often ambiguous, multiple valid branches are expected.

Multiple solutions

Constraint systems often yield multiple valid results. The `selector` callback is the main tool for choosing the subset to keep.

```
# Keep all solutions
ConstrainedArcs(..., selector=lambda arcs: arcs)

# Keep first
ConstrainedArcs(..., selector=lambda arcs: arcs[0])

# Keep shortest
ConstrainedArcs(..., selector=lambda arcs: arcs.sort_by(Edge.length)[0])
```

In Builder mode, omitting `selector` can add all solutions to context, which is often not what you want for production sketches.

Tangency qualifiers

Tangency qualifiers come from OCCT and are exposed as `Tangency`:

- `Tangency.UNQUALIFIED`: no side preference (OCCT `Unqualified`).
- `Tangency.OUTSIDE`: tangent on the exterior side of the target (OCCT `Outside`).
- `Tangency.ENCLOSING`: solution encloses/includes the target (OCCT `Enclosing`).
- `Tangency.ENCLOSED`: solution is enclosed/included by the target (OCCT `Enclosed`).

These semantics are most visible for curve-vs-curve constraints (for example circle to circle, line to circle). In many practical cases, `UNQUALIFIED` is a good default followed by filtering via `selector`.

```
with BuildLine() as egg_plant:
    # Construction Geometry
    c1 = CenterArc((-2, 0), 0.75, 80, 240, mode=Mode.PRIVATE)
    c2 = CenterArc((2, 0), 1, 220, 250, mode=Mode.PRIVATE)

    # egg_plant perimeter
    l1 = ConstrainedArcs((c2, Tangency.OUTSIDE), (c1, Tangency.OUTSIDE), radius=6)
    l2 = ConstrainedArcs(
        (c2, Tangency.ENCLOSING),
        (c1, Tangency.ENCLOSING),
        radius=8,
        selector=lambda a: a.sort_by(Axis.Y)[-1],
    )
    l3 = add(c1.trim(l1 @ 1, l2 @ 1))
    l4 = add(c2.trim(l1 @ 0, l2 @ 0))
```

In the "egg-plant" example, `Tangency.OUTSIDE` and `Tangency.ENCLOSING` reduce the candidate branches to the intended outer profile. The selector on `l2` then resolves the remaining ambiguity deterministically by choosing the highest branch in `Y`.

OCCT defines exterior/interior using orientation:

- Circle: exterior is on the right side when traversing by its orientation (interior/material is on the left side).
- Line/open curve: interior is the left side with respect to traversal direction, exterior is the opposite side.

Because of this, changing an input edge direction can change which branches satisfy `OUTSIDE/ENCLOSING/ENCLOSED`.

If qualifier behavior appears inverted, inspect input edge orientation first.

ConstrainedArcs

Overview

ConstrainedArcs supports several signature families for planar circular arcs:

1. Two tangency/contact objects + fixed radius
2. Two tangency/contact objects + center constrained on a locus
3. Three tangency/contact objects
4. One tangency/contact object + fixed center
5. One tangency/contact object + fixed radius + center constrained on a locus

`sagitta` selects short/long/both arc branches:

- `Sagitta.SHORT`
- `Sagitta.LONG`
- `Sagitta.BOTH`

In practice, use qualifiers and `sagitta` to reduce branch count, then finalize with `selector` for deterministic output.

Signature A: Two constraints + `radius`

```
ConstrainedArcs(
    tangency_one,
    tangency_two,
    radius=...,
    sagitta=Sagitta.SHORT,
    selector=lambda arcs: arcs,
)
```

Use when radius is known and arc must satisfy two contact/tangency conditions.

Signature B: Two constraints + `center_on`

```
ConstrainedArcs(
    tangency_one,
    tangency_two,
    center_on=Axis(...),  # or Edge
    sagitta=Sagitta.SHORT,
    selector=lambda arcs: arcs,
)
```

Use when center must lie on a specific line/curve rather than radius being fixed.

Signature C: Three constraints

```
ConstrainedArcs(
    tangency_one,
    tangency_two,
```

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```
    tangency_three,
    sagitta=Sagitta.BOTH,
    selector=lambda arcs: arcs,
)
```

Use for "arc tangent/contact to three entities". This can produce several branches; always consider using `selector`.

Signature D: One constraint + fixed `center`

```
ConstrainedArcs(
    tangency_one,
    center=(x, y),
    selector=lambda arcs: arcs[0],
)
```

Useful for "center-known" constructions.

Signature E: One constraint + radius + `center_on`

```
ConstrainedArcs(
    tangency_one,
    radius=...,
    center_on=some_edge,
    selector=lambda arcs: arcs,
)
```

Useful for guided-center constructions with fixed radius.

Allowed constraint objects

For arc constraints, accepted objects include:

- *Edge*
- *Axis*
- `Vertex` / `VectorLike` point
- optional qualifier wrapper: `(object, Tangency.XXX)`

ConstrainedLines

Overview

ConstrainedLines supports these signature families:

1. Tangent/contact to two objects
2. Tangent/contact to one object and passing through a fixed point
3. Tangent/contact to one object with fixed orientation (`angle` or `direction`)

Signature A: Two constraints

```
ConstrainedLines(
    tangency_one,
    tangency_two,
    selector=lambda lines: lines,
)
```

Signature B: One constraint + through point

```
ConstrainedLines(
    tangency_one,
    (x, y),  # through point
    selector=lambda lines: lines,
)
```

Signature C: One constraint + fixed orientation

```
ConstrainedLines(
    tangency_one,
    Axis.Y,
    angle=30,               # OR direction=(dx, dy)
    selector=lambda lines: lines,
)
```

Exactly one of `angle` or `direction` should be provided.

For all signatures, qualifiers can be attached to tangency inputs when side selection must be controlled.

Builder vs Algebra mode

Algebra mode

Use direct assignment and post-selection:

```
arcs = ConstrainedArcs(..., sagitta=Sagitta.BOTH)
chosen = arcs.edges().sort_by(Edge.length)[0]
```

Builder mode

Prefer selecting inside the call to avoid adding unwanted candidates to context:

```
with BuildLine() as bl:
    ConstrainedArcs(
        ...,
        sagitta=Sagitta.BOTH,
        selector=lambda arcs: arcs.sort_by(Edge.length)[0],
    )
```

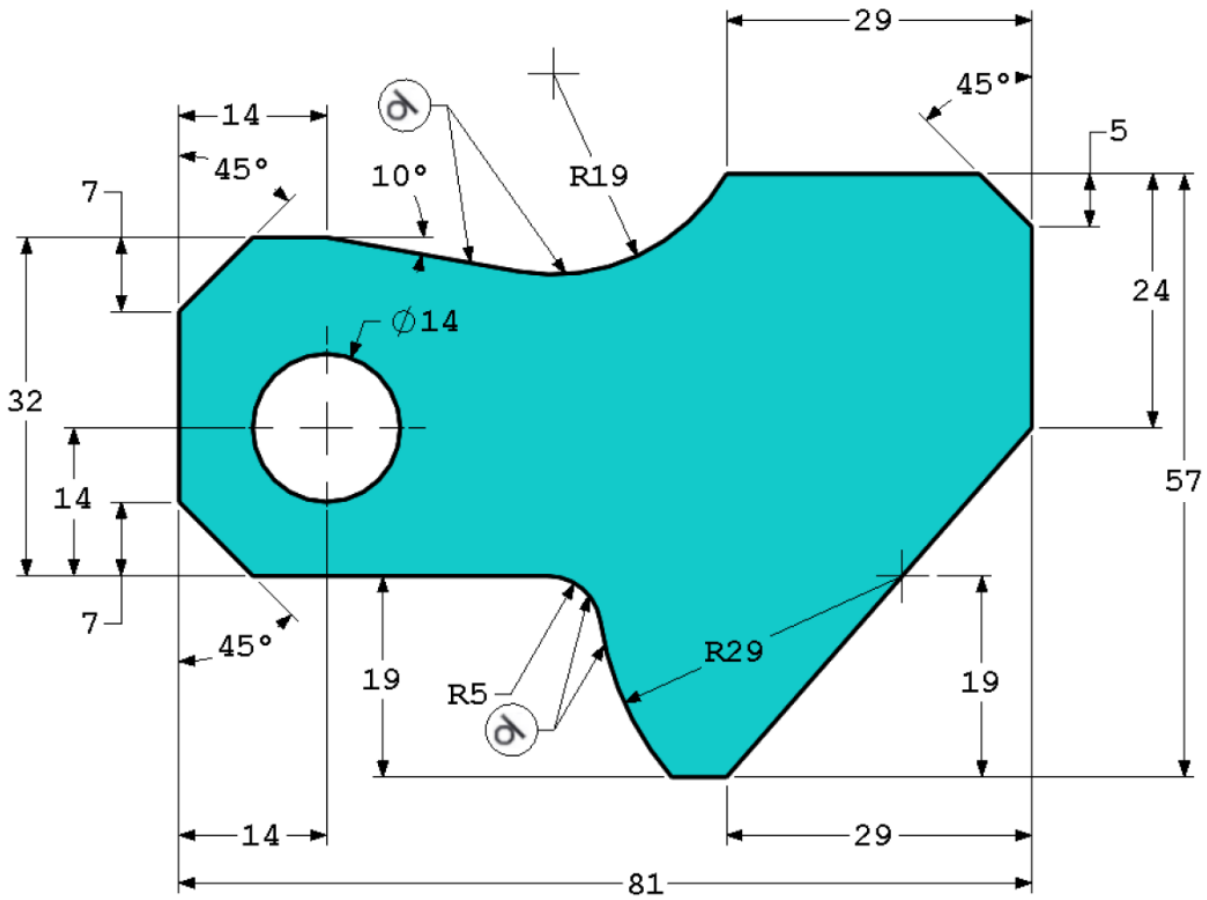
Selection recipes

```
# Nearest to point
selector=lambda edges: edges.sort_by_distance((0, 0))[0]

# Longest
selector=lambda edges: edges.sort_by(Edge.length)[-1]

# Right most
selector=lambda edges: edges.sort_by(Axis.X)[-1]

# Keep two branches
selector=lambda edges: edges[:2]
```

Prefer geometric selection criteria (distance, axis ordering, length) over positional indexing when upstream geometry may change.

Complex Drawing Example

This example pulls many of the techniques described above into a single example where the following full constrained, complex sketch is converted into build123d code.



When working with a drawing such as this one, the `ImageFace` functionality of the ocp-vscode viewer is very handy as it allows the image to be used as a visual guide when creating the sketch.

Within the following code the following conventions are used:

- construction geometry is labeled with a `c_...`
- arcs are labeled with a `a<radius>`
- lines and polylines are labeled with a `l...`

The code starts immediately above the origin (arbitrarily set to the origin of the circle) where a straight line 10° off the x-axis originates. The code then walks around the diagram clockwise creating the perimeter of the object.

```
image = ImageFace(
   "complex_sketch.png",
   scale=29 / 264,
   origin_pixels=(297, 390),
   location=Location((0, 0, -0.1)),
)

with BuildSketch() as sketch:
   with BuildLine() as perimeter:
      c_l1 = PolarLine((0, 32 - 14), 50, -10, mode=Mode.PRIVATE)
      a19 = ConstrainedArcs(c_l1, (-14 + 81 - 29, -14 - 19 + 57), radius=19)
      l2 = Polyline(a19 @ 1, a19 @ 1 + (29 - 5, 0), a19 @ 1 + (29, -5), (-14 + 81, 0))
      l3 = Line(l2 @ 1, (-14 + 81 - 29, (-14 - 19)))
      c_l4 = Line((-14, -14), (-14 + 81, -14), mode=Mode.PRIVATE)
      c_a29_arc_center = l3.intersect(c_l4)[0]
      c_a29 = CenterArc(c_a29_arc_center, 29, 180, 50, mode=Mode.PRIVATE)
      l5 = PolarLine(l3 @ 1, length=c_a29, direction=(-1, 0))
      a5 = ConstrainedArcs(
            c_a29, c_l4, radius=5, selector=lambda a: a.sort_by(Axis.X)[0]
      )
      a29 = add(c_a29.trim(l5 @ 1, a5 @ 0))
      l6 = Polyline(
            a5 @ 1,
            (-14 + 7, -14),
            (-14, -14 + 7),
            (-14, -14 + 32 - 7),
            (-14 + 7, -14 + 32),
            (0, -14 + 32),
            a19 @ 0,
      )
   make_face()
   a14 = Circle(14 / 2, mode=Mode.SUBTRACT)
```

Implementation notes:

1. Build in traversal order around the perimeter. This keeps references local and makes later edits easier because each segment depends on nearby geometry.
2. Keep helper entities private (`mode=Mode.PRIVATE`) so only final profile edges contribute to the resulting face.
3. Use named construction geometry (`c_...`) for intersections and arc centers; this improves readability and debugability.
4. Use constrained constructors only where they add value (for example *ConstrainedArcs*), and use direct primitives elsewhere.

5. Create a *Face* (`make_face` then center-hole subtraction) only after the perimeter is fully defined.

Troubleshooting

- Too many results: add qualifiers and a stricter `selector`.
- No results: relax qualifier (start with `UNQUALIFIED`) and verify geometry is coplanar.
- Unstable branch selection: avoid index-only selection when topology changes; prefer geometric sorting.
- Builder mode unexpectedly adds many edges: provide `selector` explicitly in the constructor call.

1.9.4 Lego Tutorial

This tutorial provides a step by step guide to creating a script to build a parametric Lego block as shown here:

Step 1: Setup

Before getting to the CAD operations, this Lego script needs to import the build123d environment. There are over 100 python classes in build123d so we'll just import them all with a `from build123d import *` but *there are other options* that we won't explore here.

The dimensions of the Lego block follow. A key parameter is `pip_count`, the length of the Lego blocks in pips. This parameter must be at least 2.

```
from build123d import *
from ocp_vscode import show_object
pip_count = 6

lego_unit_size = 8
pip_height = 1.8
pip_diameter = 4.8
block_length = lego_unit_size * pip_count
block_width = 16
base_height = 9.6
block_height = base_height + pip_height
support_outer_diameter = 6.5
support_inner_diameter = 4.8
ridge_width = 0.6
ridge_depth = 0.3
wall_thickness = 1.2
```

Step 2: Part Builder

The Lego block will be created by the `BuildPart` builder as it's a discrete three dimensional part; therefore, we'll instantiate a `BuildPart` with the name `lego`.

```
with BuildPart() as lego:
```

Step 3: Sketch Builder

Lego blocks have quite a bit of internal structure. To create this structure we'll draw a two dimensional sketch that will later be extruded into a three dimensional object. As this sketch will be part of the lego part, we'll create a sketch builder in the context of the part builder as follows:

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
```

Note that builder instance names are optional - we'll use `plan` to reference the sketch. Also note that all sketch objects are filled or 2D faces not just perimeter lines.

Step 4: Perimeter Rectangle

The first object in the sketch is going to be a rectangle with the dimensions of the outside of the Lego block. The following step is going to refer to this rectangle, so it will be assigned the identifier `perimeter`.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
```

Once the `Rectangle` object is created the sketch appears as follows:

Step 5: Offset to Create Walls

To create the walls of the block the rectangle that we've created needs to be hollowed out. This will be done with the `Offset` operation which is going to create a new object from `perimeter`.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
            mode=Mode.SUBTRACT,
        )
```

The first parameter to `Offset` is the reference object. The `amount` is a negative value to indicate that the offset should be internal. The `kind` parameter controls the shape of the corners - `Kind.INTERSECTION` will create square corners. Finally, the `mode` parameter controls how this object will be placed in the sketch - in this case subtracted from the existing sketch. The result is shown here:

Now the sketch consists of a hollow rectangle.

Step 6: Create Internal Grid

The interior of the Lego block has small ridges on all four internal walls. These ridges will be created as a grid of thin rectangles so the positions of the centers of these rectangles need to be defined. A pair of `GridLocations` location contexts will define these positions, one for the horizontal bars and one for the vertical bars. As the `Rectangle` objects are in the scope of a location context (`GridLocations` in this case) that defined multiple points, multiple rectangles are created.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
            mode=Mode.SUBTRACT,
        )
        # Add a grid of lengthwise and widthwise bars
        with GridLocations(x_spacing=0, y_spacing=lego_unit_size, x_count=1, y_count=2):
            Rectangle(width=block_length, height=ridge_width)
        with GridLocations(lego_unit_size, 0, pip_count, 1):
            Rectangle(width=ridge_width, height=block_width)
```

Here we can see that the first `GridLocations` creates two positions which causes two horizontal rectangles to be created. The second `GridLocations` works in the same way but creates `pip_count` positions and therefore `pip_count` rectangles. Note that keyword parameter are optional in this case.

The result looks like this:

Step 7: Create Ridges

To convert the internal grid to ridges, the center needs to be removed. This will be done with another `Rectangle`.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
            mode=Mode.SUBTRACT,
        )
        # Add a grid of lengthwise and widthwise bars
        with GridLocations(x_spacing=0, y_spacing=lego_unit_size, x_count=1, y_count=2):
            Rectangle(width=block_length, height=ridge_width)
        with GridLocations(lego_unit_size, 0, pip_count, 1):
            Rectangle(width=ridge_width, height=block_width)
        # Subtract a rectangle leaving ribs on the block walls
        Rectangle(
            block_length - 2 * (wall_thickness + ridge_depth),
            block_width - 2 * (wall_thickness + ridge_depth),
            mode=Mode.SUBTRACT,
        )
```

The `Rectangle` is subtracted from the sketch to leave the ridges as follows:

Step 8: Hollow Circles

Lego blocks use a set of internal hollow cylinders that the pips push against to hold two blocks together. These will be created with `Circle`.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
            mode=Mode.SUBTRACT,
        )
        # Add a grid of lengthwise and widthwise bars
        with GridLocations(x_spacing=0, y_spacing=lego_unit_size, x_count=1, y_count=2):
            Rectangle(width=block_length, height=ridge_width)
        with GridLocations(lego_unit_size, 0, pip_count, 1):
            Rectangle(width=ridge_width, height=block_width)
        # Subtract a rectangle leaving ribs on the block walls
        Rectangle(
            block_length - 2 * (wall_thickness + ridge_depth),
            block_width - 2 * (wall_thickness + ridge_depth),
            mode=Mode.SUBTRACT,
        )
        # Add a row of hollow circles to the center
        with GridLocations(
            x_spacing=lego_unit_size, y_spacing=0, x_count=pip_count - 1, y_count=1
        ):
            Circle(radius=support_outer_diameter / 2)
            Circle(radius=support_inner_diameter / 2, mode=Mode.SUBTRACT)
```

Here another `GridLocations` is used to position the centers of the circles. Note that since both `Circle` objects are in the scope of the location context, both Circles will be positioned at these locations.

Once the Circles are added, the sketch is complete and looks as follows:

Step 9: Extruding Sketch into Walls

Now that the sketch is complete it needs to be extruded into the three dimensional wall object.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
```

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```
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
            mode=Mode.SUBTRACT,
        )
        # Add a grid of lengthwise and widthwise bars
        with GridLocations(x_spacing=0, y_spacing=lego_unit_size, x_count=1, y_count=2):
            Rectangle(width=block_length, height=ridge_width)
        with GridLocations(lego_unit_size, 0, pip_count, 1):
            Rectangle(width=ridge_width, height=block_width)
        # Subtract a rectangle leaving ribs on the block walls
        Rectangle(
            block_length - 2 * (wall_thickness + ridge_depth),
            block_width - 2 * (wall_thickness + ridge_depth),
            mode=Mode.SUBTRACT,
        )
        # Add a row of hollow circles to the center
        with GridLocations(
            x_spacing=lego_unit_size, y_spacing=0, x_count=pip_count - 1, y_count=1
        ):
            Circle(radius=support_outer_diameter / 2)
            Circle(radius=support_inner_diameter / 2, mode=Mode.SUBTRACT)
    # Extrude this base sketch to the height of the walls
    extrude(amount=base_height - wall_thickness)
```

Note how the `Extrude` operation is no longer in the `BuildSketch` scope and has returned back into the `BuildPart` scope. This causes `BuildSketch` to exit and transfer the sketch that we've created to `BuildPart` for further processing by `Extrude`.

The result is:

Step 10: Adding a Top

Now that the walls are complete, the top of the block needs to be added. Although this could be done with another sketch, we'll add a box to the top of the walls.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
            mode=Mode.SUBTRACT,
        )
        # Add a grid of lengthwise and widthwise bars
        with GridLocations(x_spacing=0, y_spacing=lego_unit_size, x_count=1, y_count=2):
            Rectangle(width=block_length, height=ridge_width)
```

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```
        with GridLocations(lego_unit_size, 0, pip_count, 1):
            Rectangle(width=ridge_width, height=block_width)
        # Subtract a rectangle leaving ribs on the block walls
        Rectangle(
            block_length - 2 * (wall_thickness + ridge_depth),
            block_width - 2 * (wall_thickness + ridge_depth),
            mode=Mode.SUBTRACT,
        )
        # Add a row of hollow circles to the center
        with GridLocations(
            x_spacing=lego_unit_size, y_spacing=0, x_count=pip_count - 1, y_count=1
        ):
            Circle(radius=support_outer_diameter / 2)
            Circle(radius=support_inner_diameter / 2, mode=Mode.SUBTRACT)
    # Extrude this base sketch to the height of the walls
    extrude(amount=base_height - wall_thickness)
    # Create a box on the top of the walls
    with Locations((0, 0, lego.vertices().sort_by(Axis.Z)[-1].Z)):
        # Create the top of the block
        Box(
            length=block_length,
            width=block_width,
            height=wall_thickness,
            align=(Align.CENTER, Align.CENTER, Align.MIN),
        )
```

To position the top, we'll describe the top center of the lego walls with a `Locations` context. To determine the height we'll extract that from the `lego.part` by using the `vertices()` method which returns a list of the positions of all of the vertices of the Lego block so far. Since we're interested in the top, we'll sort by the vertical (Z) axis and take the top of the list `sort_by(Axis.Z)[-1]`. Finally, the `Z` property of this vertex will return just the height of the top. Note that the `X` and `Y` values are not used from the selected vertex as there are no vertices in the center of the block.

Within the scope of this `Locations` context, a `Box` is created, centered at the intersection of the x and y axis but not in the z thus aligning with the top of the walls.

The base is closed now as shown here:

Step 11: Adding Pips

The final step is to add the pips to the top of the Lego block. To do this we'll create a new workplane on top of the block where we can position the pips.

```
with BuildPart() as lego:
    # Draw the bottom of the block
    with BuildSketch() as plan:
        # Start with a Rectangle the size of the block
        perimeter = Rectangle(width=block_length, height=block_width)
        # Subtract an offset to create the block walls
        offset(
            perimeter,
            -wall_thickness,
            kind=Kind.INTERSECTION,
```

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```
            mode=Mode.SUBTRACT,
        )
        # Add a grid of lengthwise and widthwise bars
        with GridLocations(x_spacing=0, y_spacing=lego_unit_size, x_count=1, y_count=2):
            Rectangle(width=block_length, height=ridge_width)
        with GridLocations(lego_unit_size, 0, pip_count, 1):
            Rectangle(width=ridge_width, height=block_width)
        # Subtract a rectangle leaving ribs on the block walls
        Rectangle(
            block_length - 2 * (wall_thickness + ridge_depth),
            block_width - 2 * (wall_thickness + ridge_depth),
            mode=Mode.SUBTRACT,
        )
        # Add a row of hollow circles to the center
        with GridLocations(
            x_spacing=lego_unit_size, y_spacing=0, x_count=pip_count - 1, y_count=1
        ):
            Circle(radius=support_outer_diameter / 2)
            Circle(radius=support_inner_diameter / 2, mode=Mode.SUBTRACT)
    # Extrude this base sketch to the height of the walls
    extrude(amount=base_height - wall_thickness)
    # Create a box on the top of the walls
    with Locations((0, 0, lego.vertices().sort_by(Axis.Z)[-1].Z)):
        # Create the top of the block
        Box(
            length=block_length,
            width=block_width,
            height=wall_thickness,
            align=(Align.CENTER, Align.CENTER, Align.MIN),
        )
    # Create a workplane on the top of the block
    with BuildPart(lego.faces().sort_by(Axis.Z)[-1]):
        # Create a grid of pips
        with GridLocations(lego_unit_size, lego_unit_size, pip_count, 2):
            Cylinder(
                radius=pip_diameter / 2,
                height=pip_height,
                align=(Align.CENTER, Align.CENTER, Align.MIN),
            )
```

In this case, the workplane is created from the top Face of the Lego block by using the `faces` method and then sorted vertically and taking the top one `sort_by(Axis.Z)[-1]`.

On the new workplane, a grid of locations is created and a number of `Cylinder`'s are positioned at each location.

This completes the Lego block. To access the finished product, refer to the builder's internal object as shown here:

Builder	Object
BuildLine	line
BuildSketch	sketch
BuildPart	part

so in this case the Lego block is `lego.part`. To display the part use `show_object(lego.part)` or `show(lego.part)` depending on the viewer. The part could also be exported to a STL or STEP file by referencing `lego.part`.

> **Note**
>
> Viewers that don't directly support build123d my require a raw OpenCascade object. In this case, append `.wrapped` to the object (e.g.) `show_object(lego.part.wrapped)`.

1.9.5 Joint Tutorial

This tutorial provides a step by step guide in using *Joint*'s as we create a box with a hinged lid to illustrate the use of three different *Joint* types.

Step 1: Setup

Before getting to the CAD operations, this selector script needs to import the build123d environment.

```
from build123d import *
from ocp_vscode import *
```

Step 2: Create Hinge

This example uses a common Butt Hinge to connect the lid to the box base so a `Hinge` class is used to create that can create either of the two hinge leaves. As the focus of this tutorial is the joints and not the CAD operations to create objects, this code is not described in detail.

```
class Hinge(Compound):
    """Hinge

    Half a simple hinge with several joints. The joints are:
    - "leaf": RigidJoint where hinge attaches to object
    - "hinge_axis": RigidJoint (inner) or RevoluteJoint (outer)
    - "hole0", "hole1", "hole2": CylindricalJoints for attachment screws

    Args:
        width (float): width of one leaf
        length (float): hinge length
        barrel_diameter (float): size of hinge pin barrel
        thickness (float): hinge leaf thickness
        pin_diameter (float): hinge pin diameter
        inner (bool, optional): inner or outer half of hinge . Defaults to True.
    """

    def __init__(
        self,
        width: float,
        length: float,
        barrel_diameter: float,
        thickness: float,
        pin_diameter: float,
        inner: bool = True,
```

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```
    ):
        # The profile of the hinge used to create the tabs
        with BuildPart() as hinge_profile:
            with BuildSketch():
                for i, loc in enumerate(
                    GridLocations(0, length / 5, 1, 5, align=(Align.MIN, Align.MIN))
                ):
                    if i % 2 == inner:
                        with Locations(loc):
                            Rectangle(width, length / 5, align=(Align.MIN, Align.MIN))
                Rectangle(
                    width - barrel_diameter,
                    length,
                    align=(Align.MIN, Align.MIN),
                )
            extrude(amount=-barrel_diameter)

        # The hinge pin
        with BuildPart() as pin:
            Cylinder(
                radius=pin_diameter / 2,
                height=length,
                align=(Align.CENTER, Align.CENTER, Align.MIN),
            )
            with BuildPart(pin.part.faces().sort_by(Axis.Z)[-1]) as pin_head:
                Cylinder(
                    radius=barrel_diameter / 2,
                    height=pin_diameter,
                    align=(Align.CENTER, Align.CENTER, Align.MIN),
                )
            fillet(
                pin_head.edges(Select.LAST).filter_by(GeomType.CIRCLE),
                radius=pin_diameter / 3,
            )

        # Either the external and internal leaf with joints
        with BuildPart() as leaf_builder:
            with BuildSketch():
                with BuildLine():
                    l1 = Line((0, 0), (width - barrel_diameter / 2, 0))
                    l2 = RadiusArc(
                        l1 @ 1,
                        l1 @ 1 + Vector(0, barrel_diameter),
                        -barrel_diameter / 2,
                    )
                    l3 = RadiusArc(
                        l2 @ 1,
                        (
                            width - barrel_diameter,
                            barrel_diameter / 2,
                        ),
                        -barrel_diameter / 2,
```

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```
                )
                l4 = Line(l3 @ 1, (width - barrel_diameter, thickness))
                l5 = Line(l4 @ 1, (0, thickness))
                Line(l5 @ 1, l1 @ 0)
            make_face()
            with Locations(
                (width - barrel_diameter / 2, barrel_diameter / 2)
            ) as pin_center:
                Circle(pin_diameter / 2 + 0.1 * MM, mode=Mode.SUBTRACT)
        extrude(amount=length)
        add(hinge_profile.part, rotation=(90, 0, 0), mode=Mode.INTERSECT)

        # Create holes for fasteners
        with Locations(leaf_builder.part.faces().filter_by(Axis.Y)[-1]):
            with GridLocations(0, length / 3, 1, 3):
                holes = CounterSinkHole(3 * MM, 5 * MM)
        # Add the hinge pin to the external leaf
        if not inner:
            with Locations(pin_center.locations[0]):
                add(pin.part)
```

Once the two leaves have been created they will look as follows:

Note that the XYZ indicators and a circle around the hinge pin indicate joints that are discussed below.

Step 3: Add Joints to the Hinge Leaf

The hinge includes five joints:

- A `RigidJoint` to attach the leaf
- A `RigidJoint` or `RevoluteJoint` as the hinge Axis
- Three `CylindricalJoint`'s for the countersunk screws

Step 3a: Leaf Joint

The first joint to add is a `RigidJoint` that is used to fix the hinge leaf to the box or lid.

```
        #
        # Leaf attachment
        RigidJoint(
            label="leaf",
            joint_location=Location(
                (width - barrel_diameter, 0, length / 2), (90, 0, 0)
            ),
        )
```

Each joint has a label which identifies it - here the string "leaf" is used, the `to_part` binds the joint to `leaf_builder.part` (i.e. the part being built), and `joint_location` is specified as middle of the leaf along the edge of the pin. Note that *`Location`* objects describe both a position and orientation which is why there are two tuples (the orientation listed is rotate about the X axis 90 degrees).

Step 3b: Hinge Joint

The second joint to add is either a `RigidJoint` (on the inner leaf) or a `RevoluteJoint` (on the outer leaf) that describes the hinge axis.

```
            #
            # Leaf attachment
            RigidJoint(
                label="leaf",
                joint_location=Location(
                    (width - barrel_diameter, 0, length / 2), (90, 0, 0)
                ),
            )
            # [Hinge Axis] (fixed with inner)
            if inner:
                RigidJoint(
                    "hinge_axis",
                    joint_location=Location(
                        (width - barrel_diameter / 2, barrel_diameter / 2, 0)
                    ),
                )
            else:
                RevoluteJoint(
                    "hinge_axis",
                    axis=Axis(
                        (width - barrel_diameter / 2, barrel_diameter / 2, 0), (0, 0, 1)
                    ),
                    angular_range=(90, 270),
                )
```

The inner leaf just pivots around the outer leaf and therefore the simple `RigidJoint` is used to define the Location of this pivot. The outer leaf contains the more complex `RevoluteJoint` which defines an axis of rotation and angular limits to that rotation (90 and 270 in this example as the two leaves will interfere with each other outside of this range). Note that the maximum angle must be greater than the minimum angle and therefore may be greater than 360°. Other types of joints have linear ranges as well as angular ranges.

Step 3c: Fastener Joints

The third set of joints to add are `CylindricalJoint`'s that describe how the countersunk screws used to attach the leaves move.

```
            hole_locations = [hole.location for hole in holes]
            for hole, hole_location in enumerate(hole_locations):
                CylindricalJoint(
                    label="hole" + str(hole),
                    axis=Axis(hole_location),
                    linear_range=(-2 * CM, 2 * CM),
                    angular_range=(0, 360),
                )
```

Much like the `RevoluteJoint`, a `CylindricalJoint` has an Axis of motion but this type of joint allows both movement around and along this axis - exactly as a screw would move. Here is the Axis is setup such that a position of 0 aligns with the screw being fully set in the hole and positive numbers indicate the distance the head of the screw is above the leaf surface. One could have reversed the direction of the Axis such that negative position values would correspond to a screw now fully in the hole - whatever makes sense to the situation. The angular range of this joint is

set to (0°, 360°) as there is no limit to the angular rotation of the screw (one could choose to model thread pitch and calculate position from angle or vice-versa).

Step 3d: Call Super

To finish off, the base class for the Hinge class is initialized:

```
        super().__init__(leaf_builder.part.wrapped, joints=leaf_builder.part.joints)
```

Step 3e: Instantiate Hinge Leaves

Now that the Hinge class is complete it can be used to instantiate the two hinge leaves required to attach the box and lid together.

```
hinge_inner = Hinge(
    width=5 * CM,
    length=12 * CM,
    barrel_diameter=1 * CM,
    thickness=2 * MM,
    pin_diameter=4 * MM,
)
hinge_outer = Hinge(
    width=5 * CM,
    length=12 * CM,
    barrel_diameter=1 * CM,
    thickness=2 * MM,
    pin_diameter=4 * MM,
    inner=False,
)
```

Step 4: Create the Box

The box is created with *BuildPart* as a simple object - as shown below - let's focus on the joint used to attach the outer hinge leaf.

```
with BuildPart() as box_builder:
    box = Box(30 * CM, 30 * CM, 10 * CM)
    offset(amount=-1 * CM, openings=box_builder.faces().sort_by(Axis.Z)[-1])
    # Create a notch for the hinge
    with Locations((-15 * CM, 0, 5 * CM)):
        Box(2 * CM, 12 * CM, 4 * MM, mode=Mode.SUBTRACT)
    bbox = box.bounding_box()
    with Locations(
        Plane(origin=(bbox.min.X, 0, bbox.max.Z - 30 * MM), z_dir=(-1, 0, 0))
    ):
        with GridLocations(0, 40 * MM, 1, 3):
            Hole(3 * MM, 1 * CM)
    RigidJoint(
        "hinge_attachment",
        joint_location=Location((-15 * CM, 0, 4 * CM), (180, 90, 0)),
    )
```

Since the hinge will be fixed to the box another `RigidJoint` is used mark where the hinge will go. Note that the orientation of this *Joint* will control how the hinge leaf is attached and is independent of the orientation of the hinge as it was constructed.

Step 4a: Relocate Box

Note that the position and orientation of the box's joints are given as a global *Location* when created but will be translated to a relative *Location* internally to allow the *Joint* to "move" with the parent object. This allows users the freedom to relocate objects without having to recreate or modify *Joint*'s. Here is the box is moved upwards to show this property.

```
box = box_builder.part.moved(Location((0, 0, 5 * CM)))
```

Step 5: Create the Lid

Much like the box, the lid is created in a *BuildPart* context and is assigned a `RigidJoint`.

```
with BuildPart() as lid_builder:
    Box(30 * CM, 30 * CM, 1 * CM, align=(Align.MIN, Align.CENTER, Align.MIN))
    with Locations((2 * CM, 0, 0)):
        with GridLocations(0, 40 * MM, 1, 3):
            Hole(3 * MM, 1 * CM)
    RigidJoint(
        "hinge_attachment",
        joint_location=Location((0, 0, 0), (0, 0, 180)),
    )
lid = lid_builder.part
```

Again, the original orientation of the lid and hinge inner leaf are not important, when the joints are connected together the parts will move into the correct position.

Step 6: Import a Screw and bind a Joint to it

Joint's can be bound to simple objects the a *Compound* imported - in this case a screw.

- screw STEP model: `M6-1x12-countersunk-screw.step`

```
m6_screw = import_step("M6-1x12-countersunk-screw.step")
m6_joint = RigidJoint("head", m6_screw, Location((0, 0, 0), (0, 0, 0)))
```

Here a simple `RigidJoint` is bound to the top of the screw head such that it can be connected to the hinge's `CylindricalJoint`.

Step 7: Connect the Joints together

This last step is the most interesting. Now that all of the joints have been defined and bound to their parent objects, they can be connected together.

Step 7a: Hinge to Box

To start, the outer hinge leaf will be connected to the box, as follows:

```
box.joints["hinge_attachment"].connect_to(hinge_outer.joints["leaf"])
```

Here the `hinge_attachment` joint of the `box` is connected to the `leaf` joint of `hinge_outer`. Note that the hinge leaf is the object to move. Once this line is executed, we get the following:

Step 7b: Hinge to Hinge

Next, the hinge inner leaf is connected to the hinge outer leaf which is attached to the box.

```
hinge_outer.joints["hinge_axis"].connect_to(hinge_inner.joints["hinge_axis"], angle=120)
```

As `hinge_outer.joints["hinge_axis"]` is a `RevoluteJoint` there is an `angle` parameter that can be set (angles default to the minimum range value) - here to 120°. This is what that looks like:

Step 7c: Lid to Hinge

Now the `lid` is connected to the `hinge_inner`:

```
hinge_inner.joints["leaf"].connect_to(lid.joints["hinge_attachment"])
```

which results in:

Note how the lid is now in an open position. To close the lid just change the above `angle` parameter from 120° to 90°.

Step 7d: Screw to Hinge

The last step in this example is to place a screw in one of the hinges:

```
hinge_outer.joints["hole2"].connect_to(m6_joint, position=5 * MM, angle=30)
```

As the position is a positive number the screw is still proud of the hinge face as shown here:

Try changing these position and angle values to "tighten" the screw.

Conclusion

Use a *Joint* to locate two objects relative to each other with some degree of motion. Keep in mind that when using the `connect_to` method, `self` is always fixed and `other` will move to the appropriate *Location*.

Note

The joint symbols can be displayed as follows (your viewer may use `show` instead of `show_object`):

```
show_object(box.joints["hinge_attachment"].symbol, name="box attachment point")
```

or

```
show_object(m6_joint.symbol, name="m6 screw symbol")
```

or, with the ocp_vscode viewer

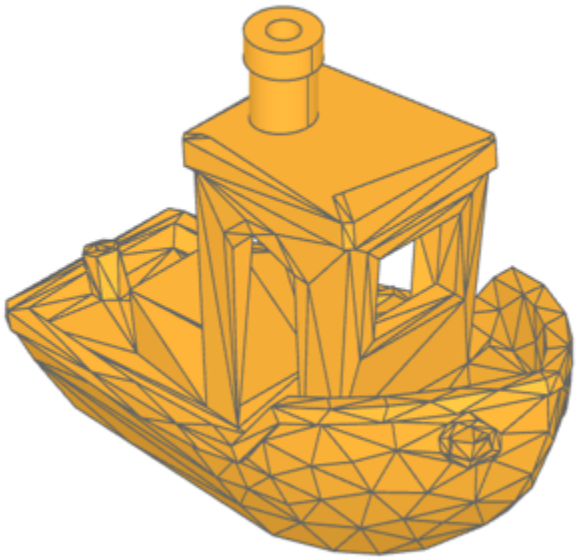
```
show(box, render_joints=True)
```

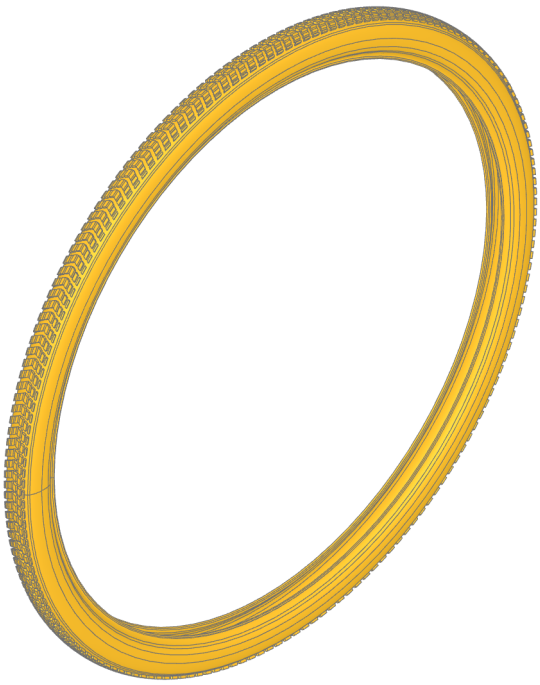
1.9.6 The build123d Examples

Overview

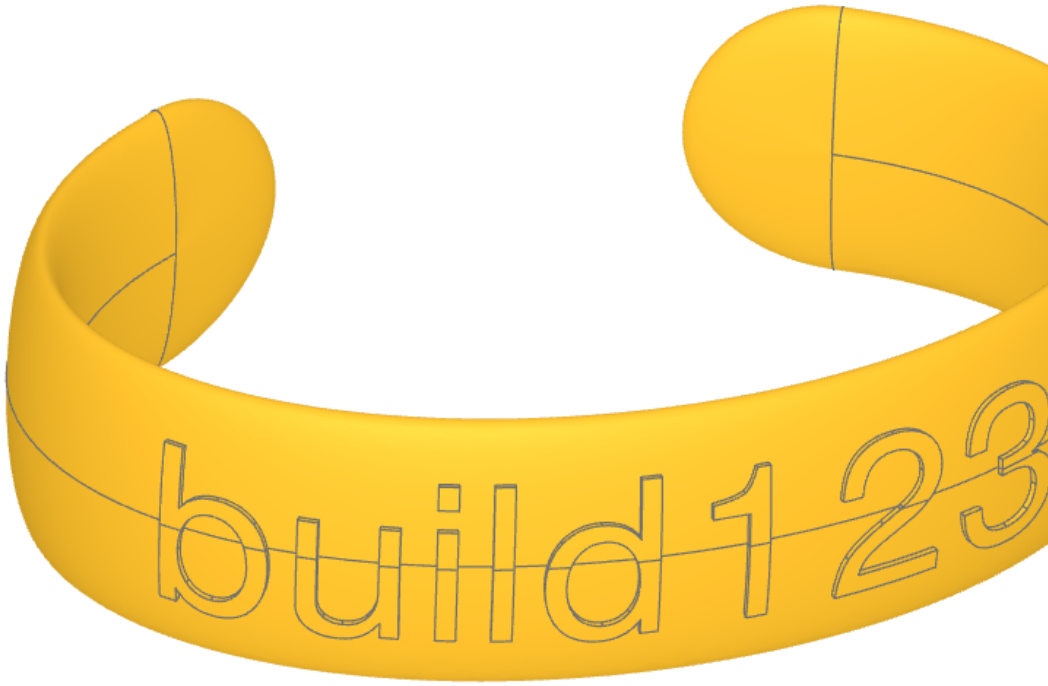
In the GitHub repository you will find an examples folder.

Most of the examples show the builder and algebra modes.





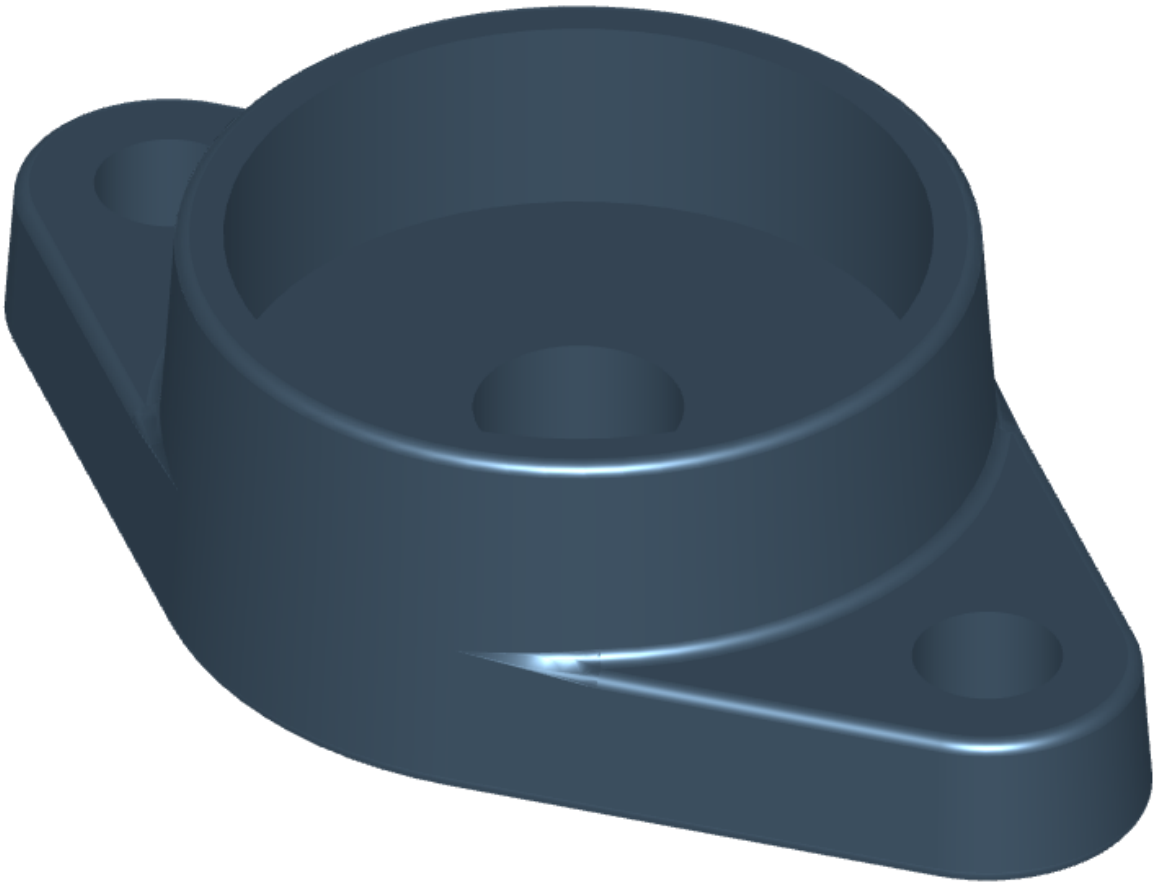
Benchy *Benchy*

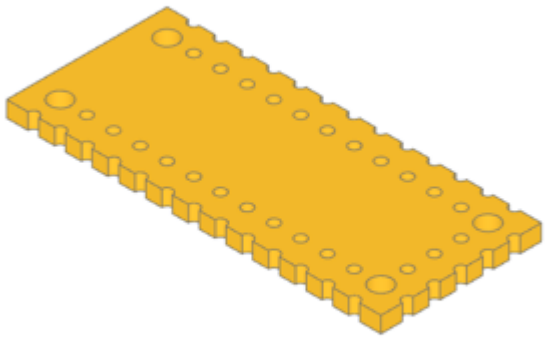


Bicycle Tire *Bicycle Tire*

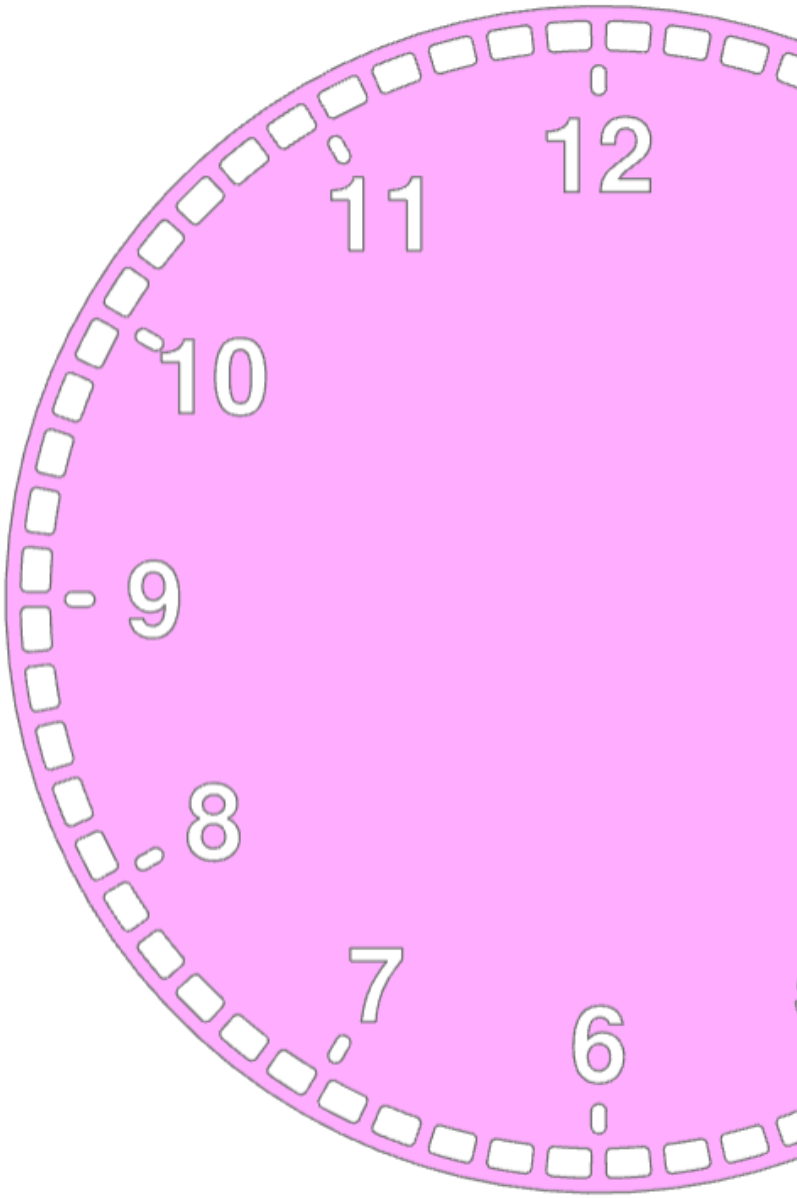
Bracelet *Bracelet*

Canadian Flag Blowing in The Wind *Canadian Flag Blowing in The Wind*

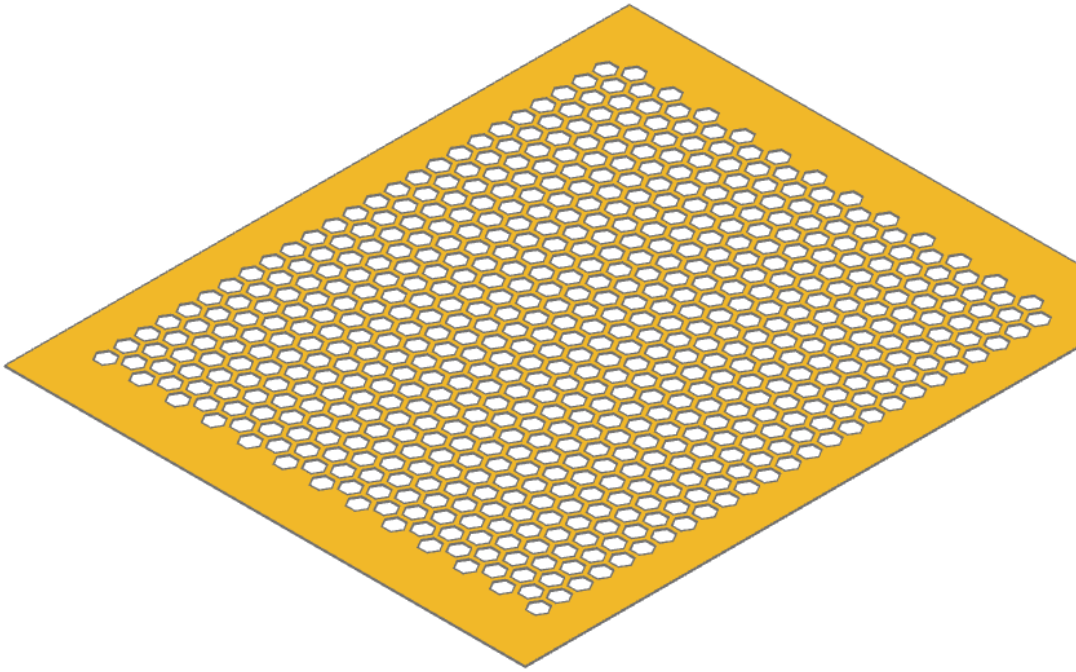




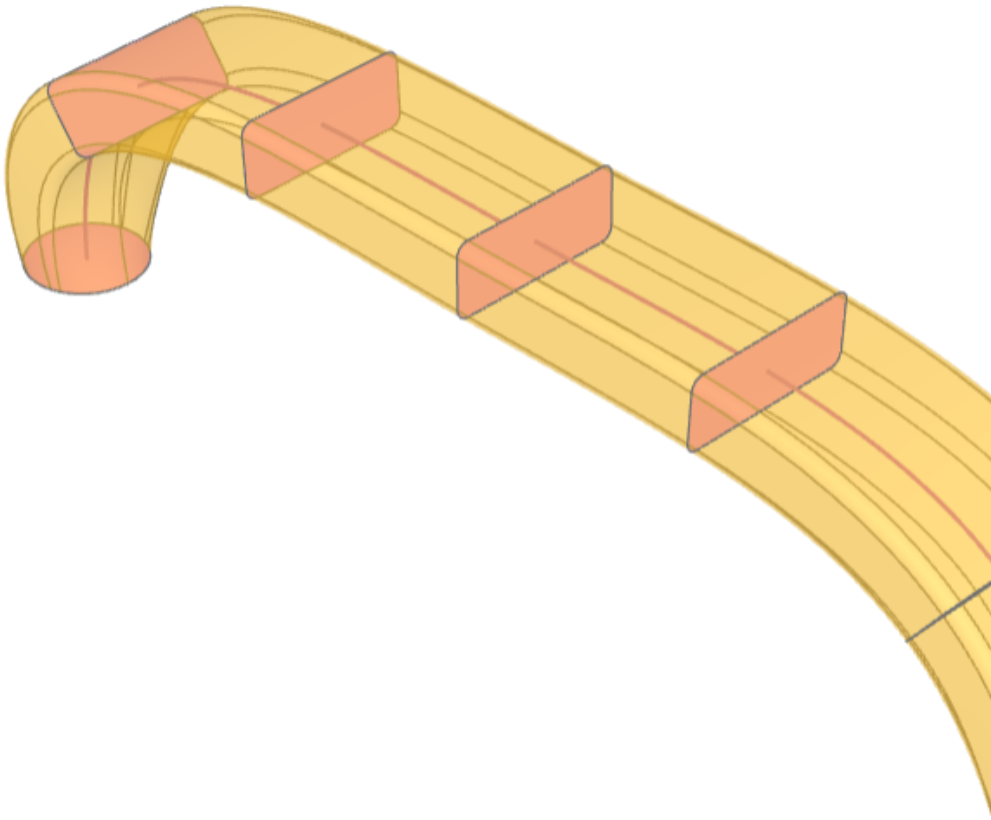
Cast Bearing Unit *Cast Bearing Unit*



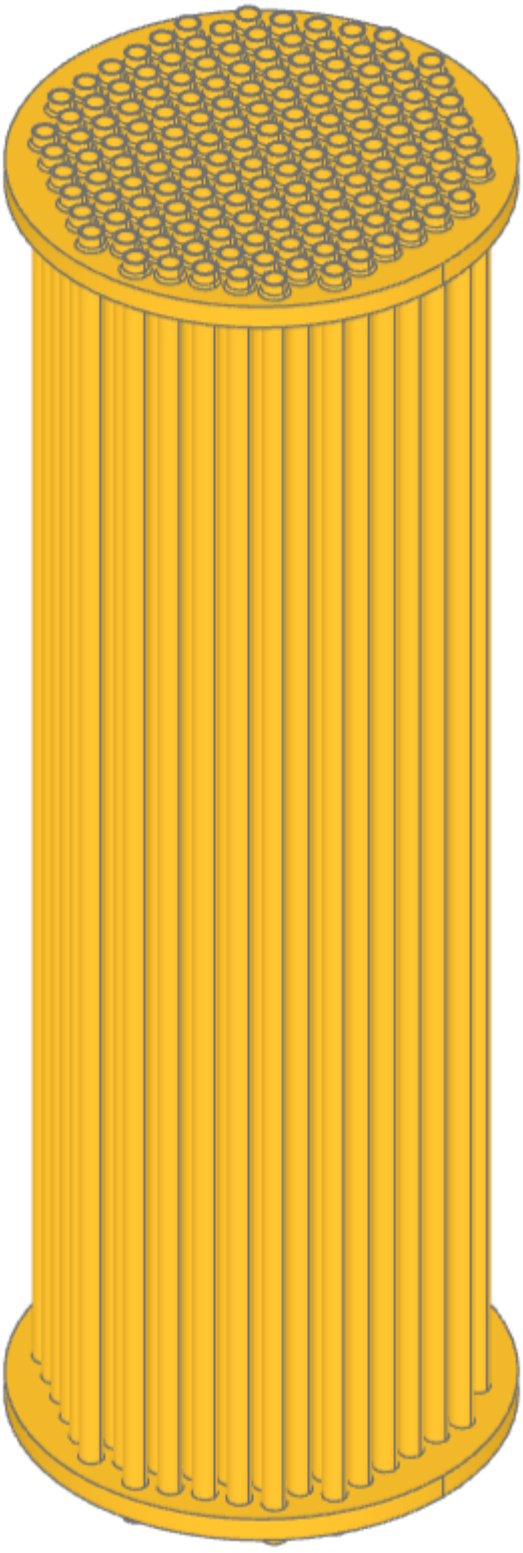
Circuit Board With Holes *Circuit Board With Holes*



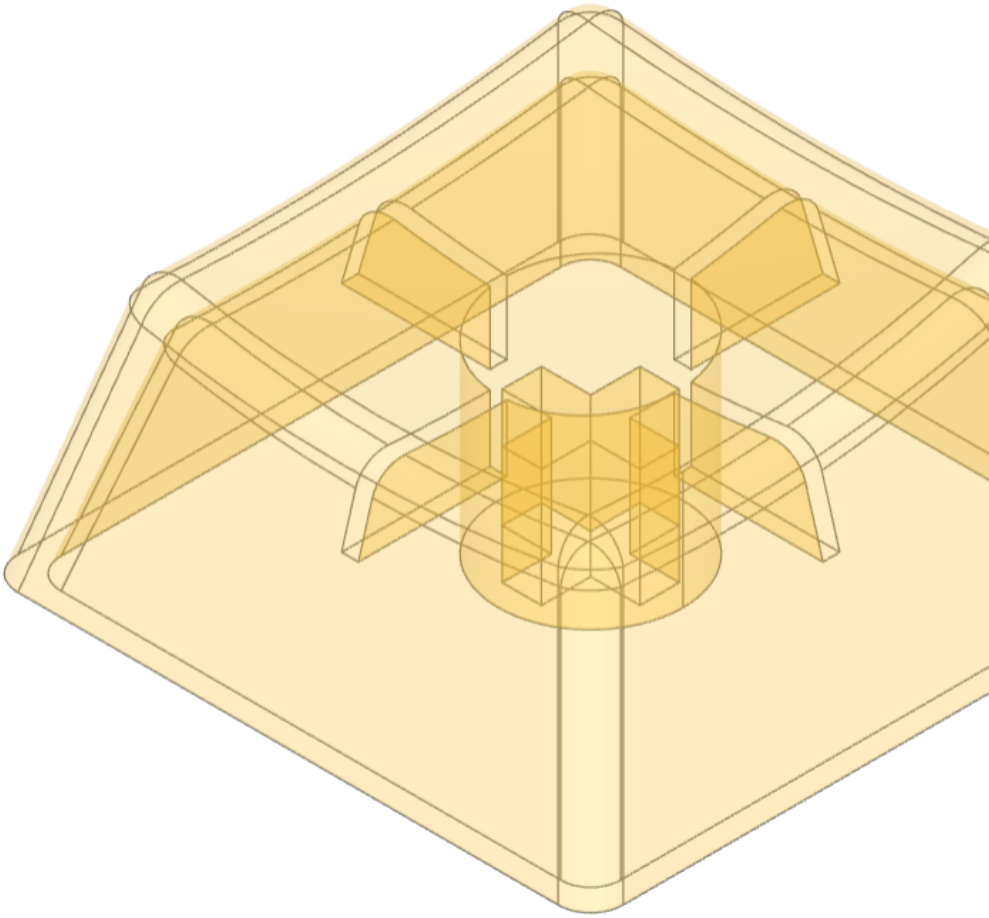
Clock Face *Clock Face*



Fast Grid Holes *Fast Grid Holes*

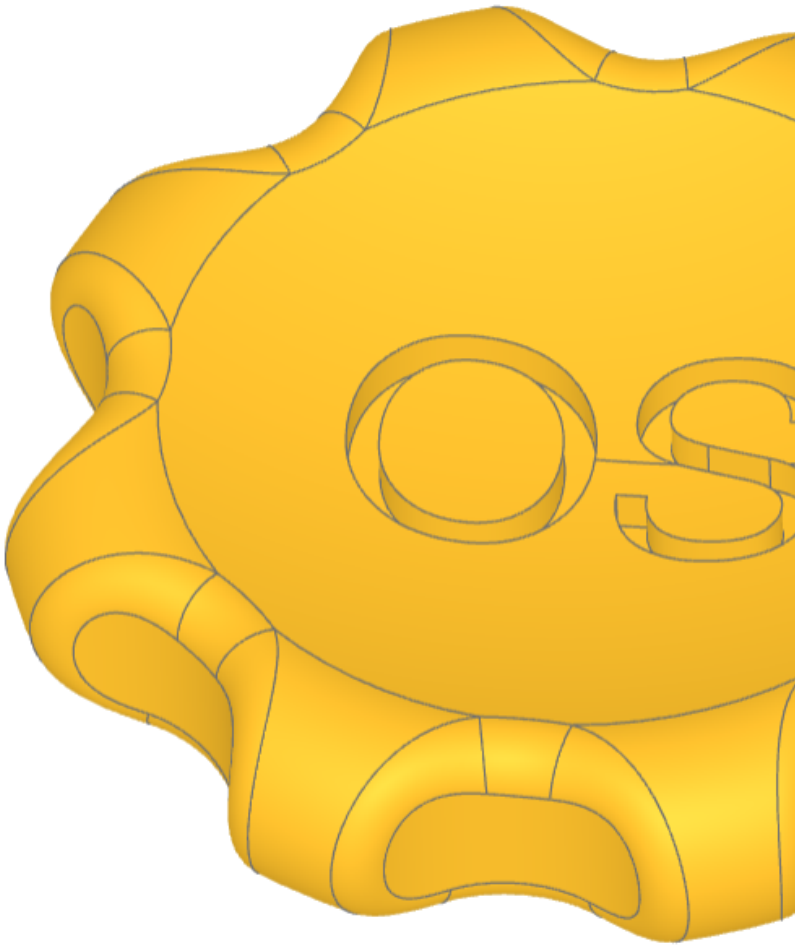


Handle *Handle*

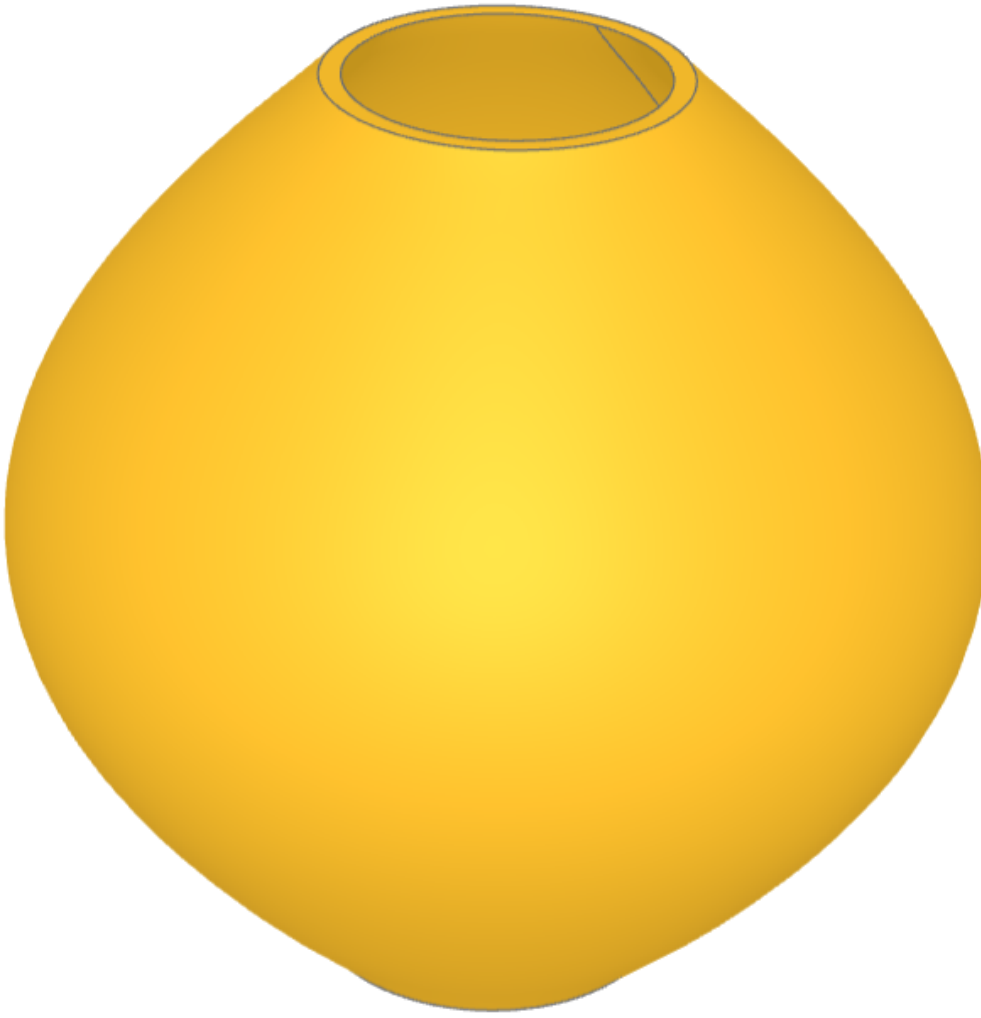


Heat Exchanger *Heat Exchanger*

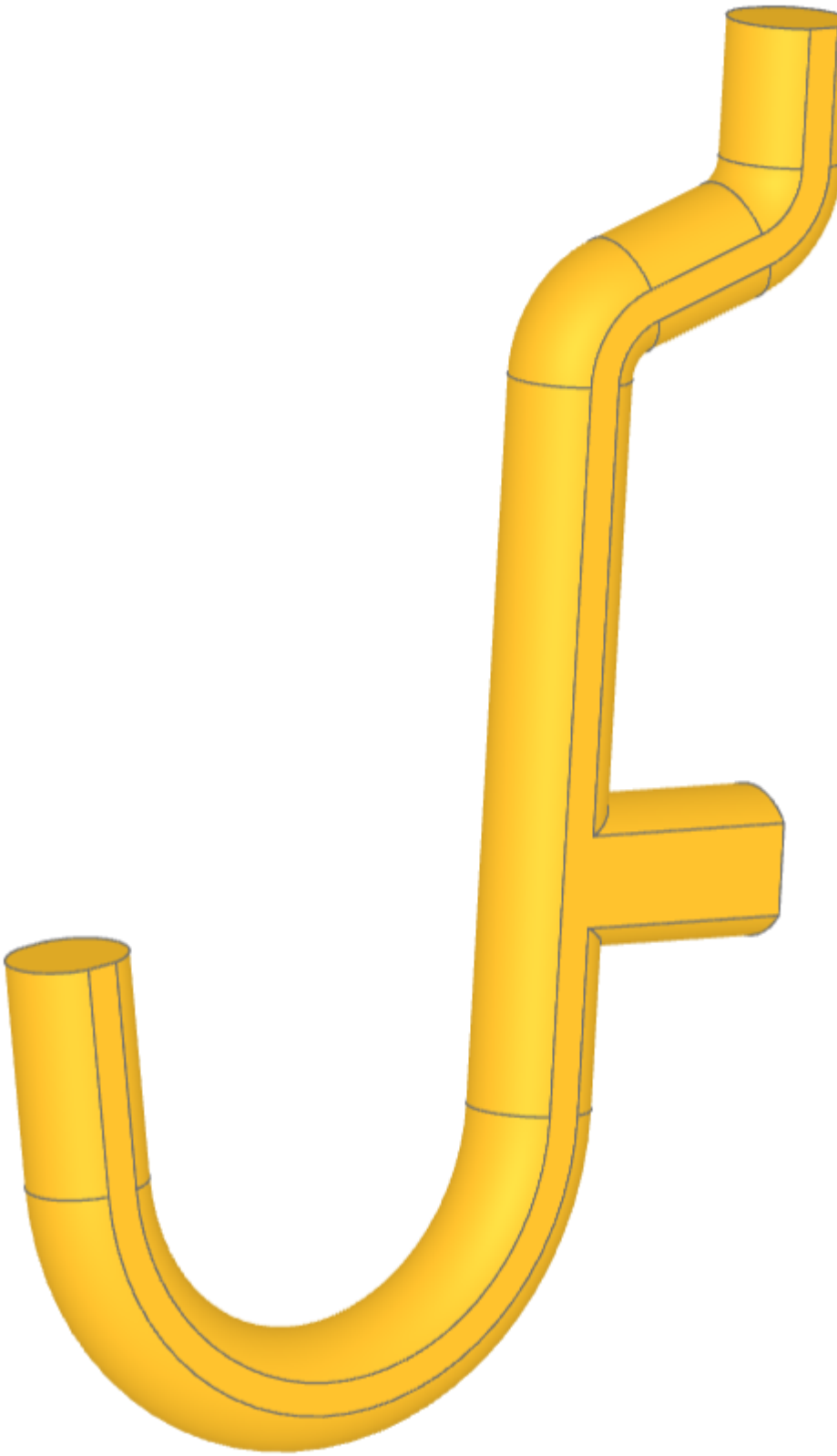
Key Cap *Key Cap*



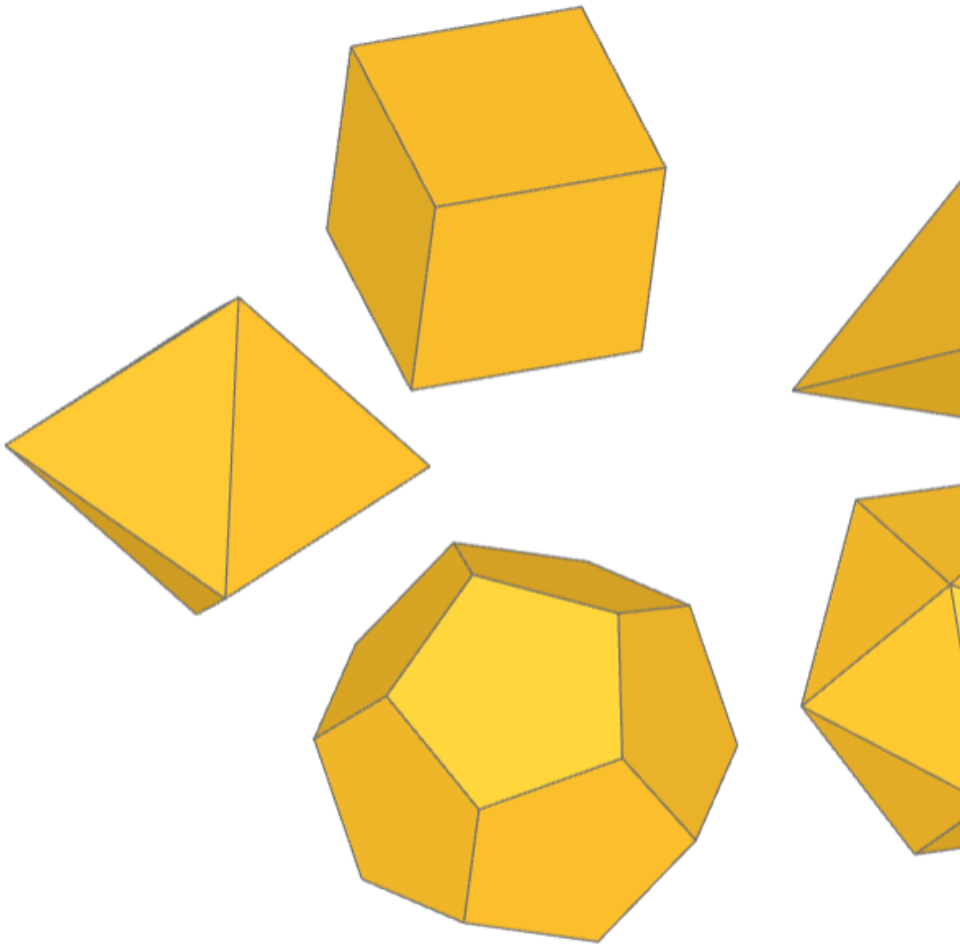
(former) build123d Logo *Former build123d Logo*



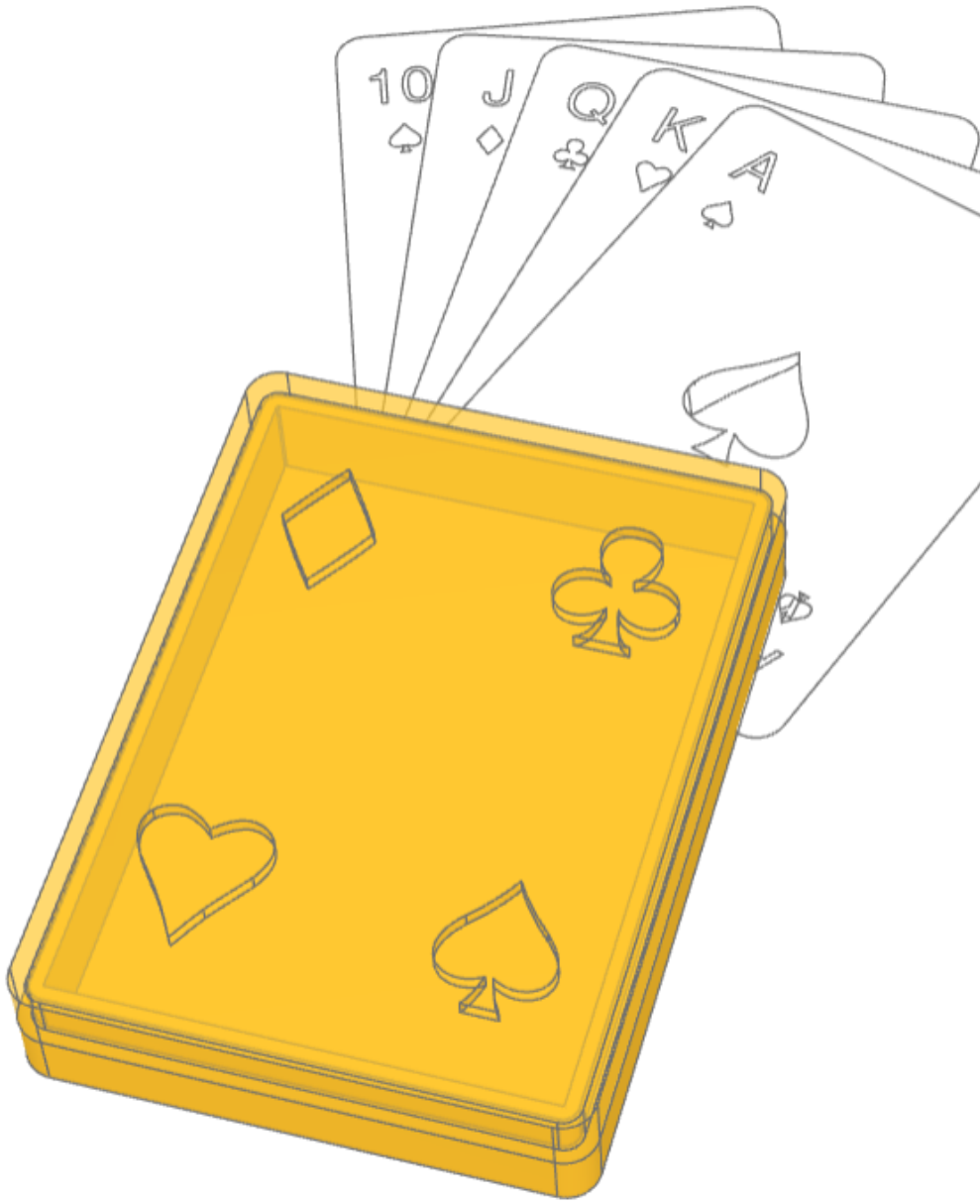
Maker Coin *Maker Coin*



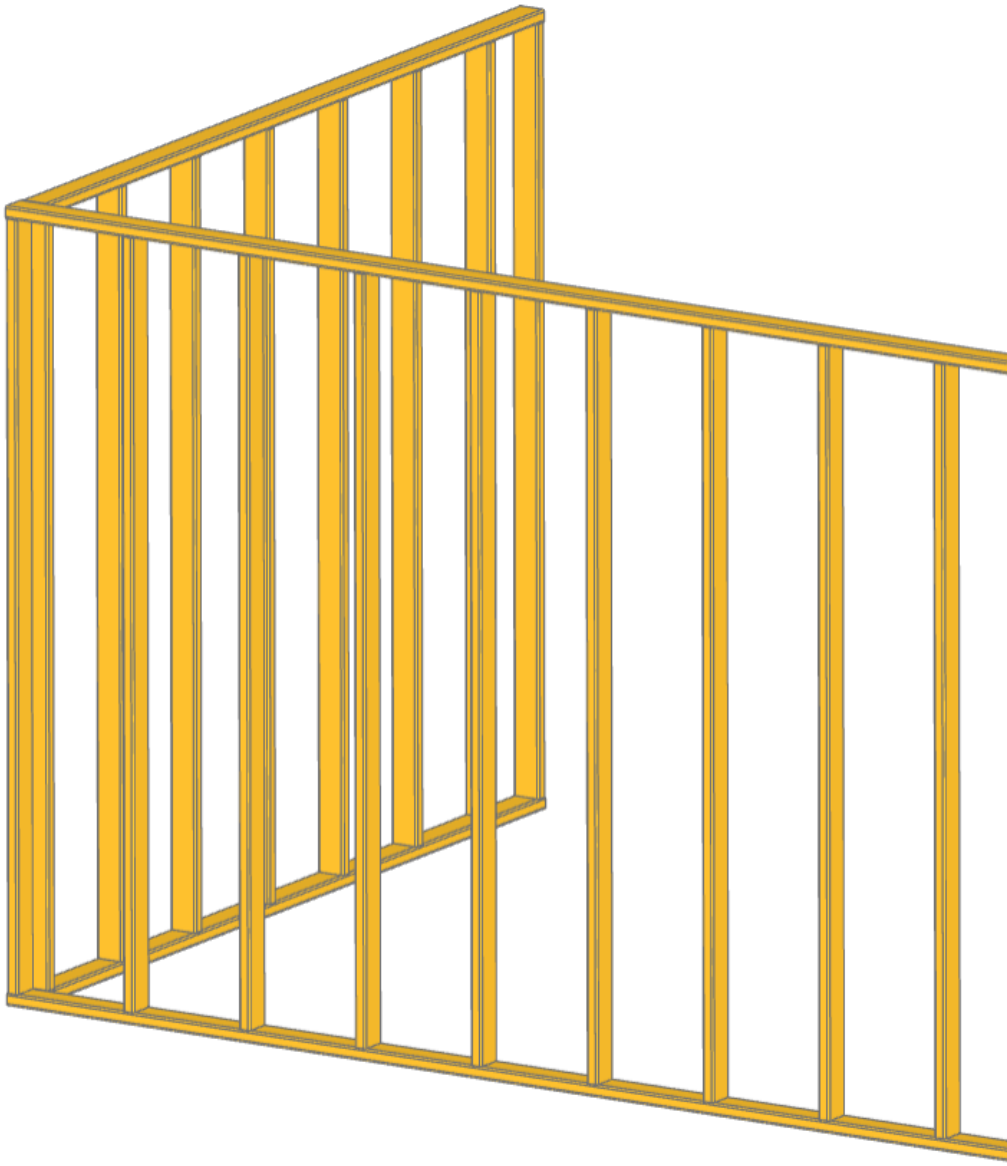
Multi-Sketch Loft *Multi-Sketch Loft*



Peg Board J Hook *Peg Board Hook*

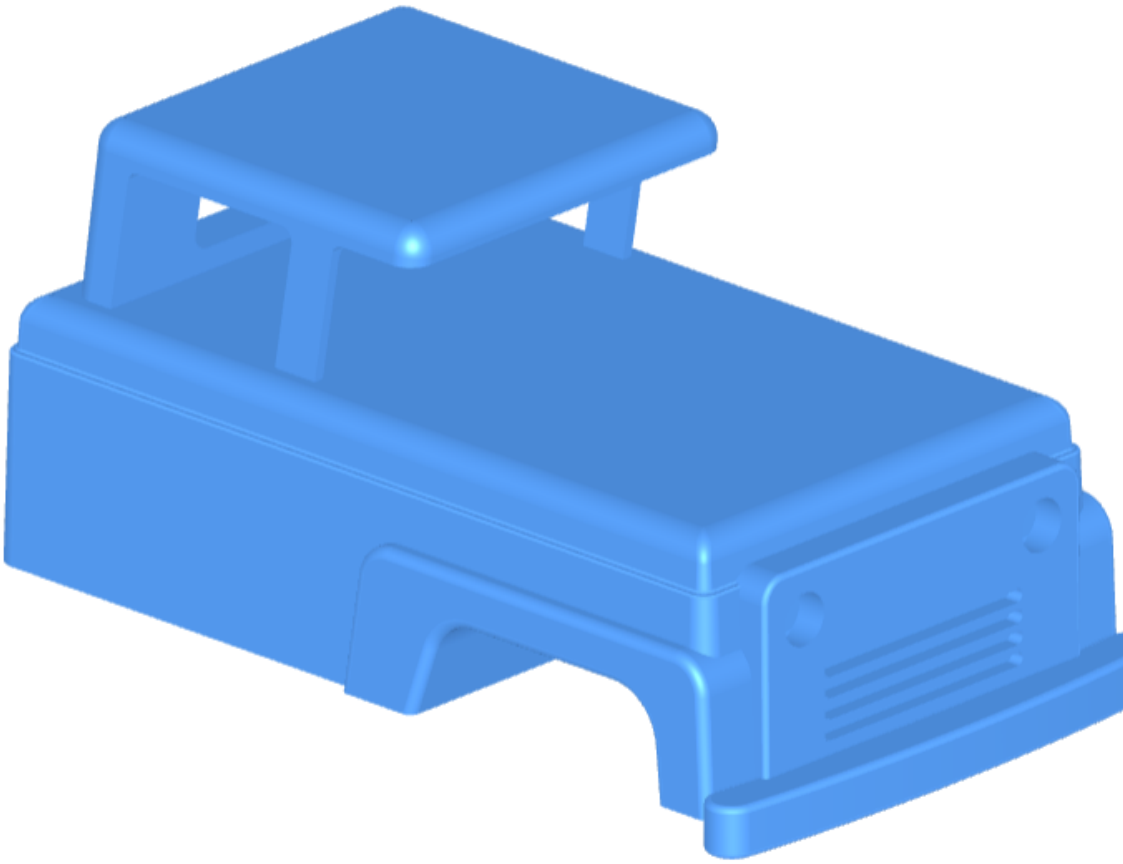


Platonic Solids *Platonic Solids*



Playing Cards *Playing Cards*

Stud Wall *Stud Wall*

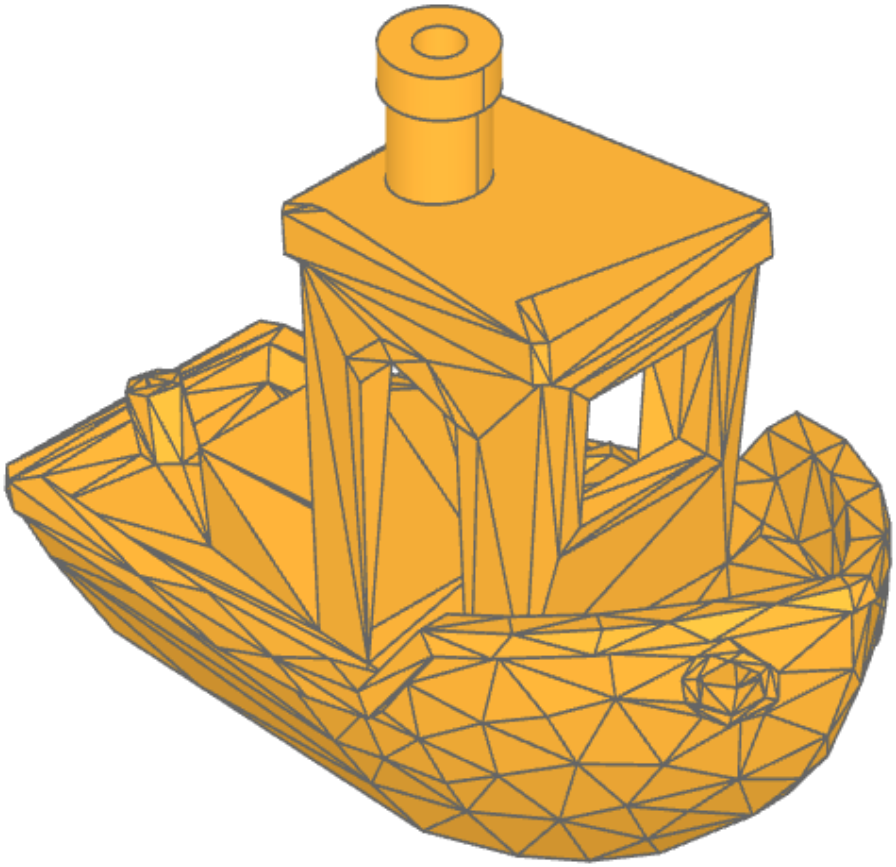


Tea Cup *Tea Cup*

Toy Truck *Toy Truck*

Vase *Vase*

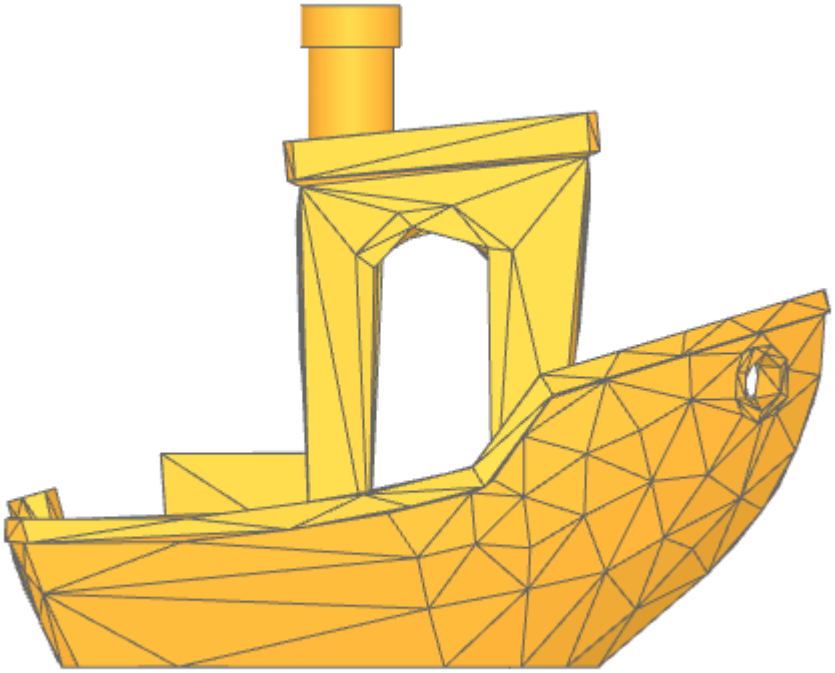
Benchy

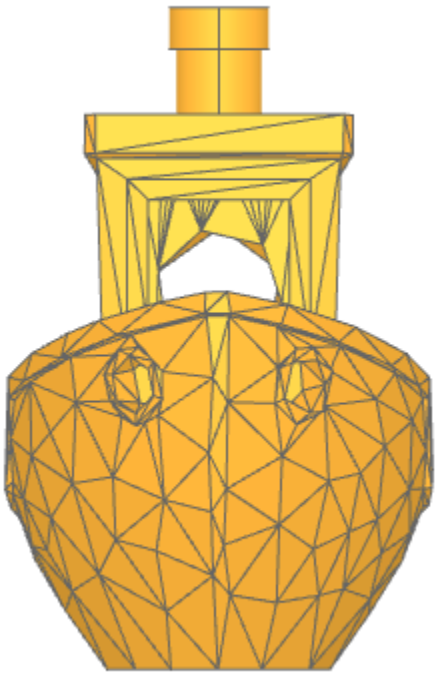


The Benchy examples shows how to import a STL model as a *Solid* object with the class *Mesher* and modify it by replacing chimney with a BREP version.

- Benchy STL model: `low_poly_benchy.stl`

Gallery





Reference Implementation (Builder Mode)

```
# Import the benchy as a Solid model
importer = Mesher()
benchy_stl = importer.read("low_poly_benchy.stl")[0]

with BuildPart() as benchy:
    add(benchy_stl)

    # Determine the plane that defines the top of the roof
    vertices = benchy.vertices()
    roof_vertices = vertices.filter_by_position(Axis.Z, 38, 42)
    roof_plane_vertices = [
        roof_vertices.group_by(Axis.Y, tol_digits=2)[-1].sort_by(Axis.X)[0],
        roof_vertices.sort_by(Axis.Z)[0],
        roof_vertices.group_by(Axis.Y, tol_digits=2)[0].sort_by(Axis.X)[0],
    ]
    roof_plane = Plane(
        Face(Wire.make_polygon([v.to_tuple() for v in roof_plane_vertices]))
    )
    # Remove the faceted smoke stack
    split(bisect_by=roof_plane, keep=Keep.BOTTOM)

    # Determine the position and size of the smoke stack
    smoke_stack_vertices = vertices.group_by(Axis.Z, tol_digits=0)[-1]
```

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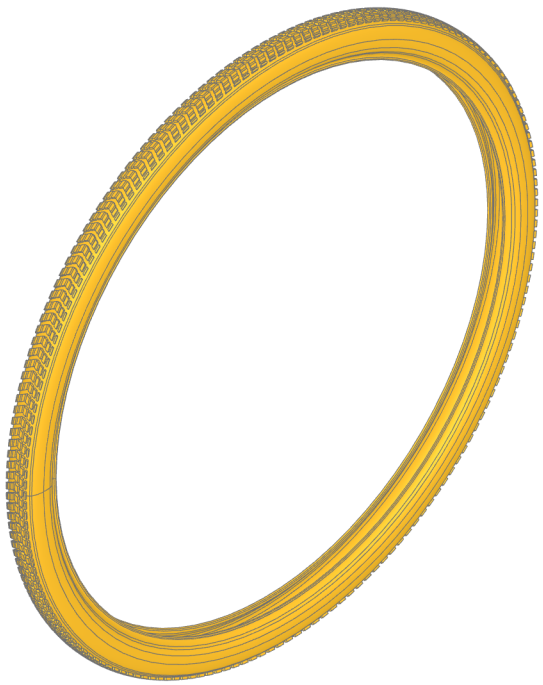
```
    smoke_stack_center = sum(
        [Vector(v.X, v.Y, v.Z) for v in smoke_stack_vertices], Vector()
    ) * (1 / len(smoke_stack_vertices))
    smoke_stack_radius = max(
        [
            (Vector(*v.to_tuple()) - smoke_stack_center).length
            for v in smoke_stack_vertices
        ]
    )

    # Create the new smoke stack
    with BuildSketch(Plane(smoke_stack_center)):
        Circle(smoke_stack_radius)
        Circle(smoke_stack_radius - 2 * MM, mode=Mode.SUBTRACT)
    extrude(amount=-3 * MM)
    with BuildSketch(Plane(smoke_stack_center)):
        Circle(smoke_stack_radius - 0.5 * MM)
        Circle(smoke_stack_radius - 2 * MM, mode=Mode.SUBTRACT)
    extrude(amount=roof_plane_vertices[1].Z - smoke_stack_center.Z)

show(benchy)
```

Bicycle Tire



This example demonstrates how to model a realistic bicycle tire with a patterned tread using build123d. The key concept showcased here is the use of wrap_faces to project 2D planar geometry onto a curved 3D surface.

Reference Implementation (Builder Mode)

```
import copy
from build123d import *
from ocp_vscode import show

wheel_diameter = 740 * MM

with BuildSketch() as tire_profile:
    with BuildLine() as build_profile:
        l00 = Bezier((0.0, 0.0), (7.05, 0.0), (12.18, 1.54), (15.13, 4.54))
        l01 = Bezier(l00 @ 1, (15.81, 5.22), (15.98, 5.44), (16.5, 6.23))
        l02 = Bezier(l01 @ 1, (18.45, 9.19), (19.61, 13.84), (19.94, 20.06))
        l03 = Bezier(l02 @ 1, (20.1, 23.24), (19.93, 27.48), (19.56, 29.45))
        l04 = Bezier(l03 @ 1, (19.13, 31.69), (18.23, 33.67), (16.91, 35.32))
        l05 = Bezier(l04 @ 1, (16.26, 36.12), (15.57, 36.77), (14.48, 37.58))
        l06 = Bezier(l05 @ 1, (12.77, 38.85), (11.51, 40.28), (10.76, 41.78))
        l07 = Bezier(l06 @ 1, (10.07, 43.16), (10.15, 43.81), (11.03, 43.98))
        l08 = Bezier(l07 @ 1, (11.82, 44.13), (12.15, 44.55), (12.08, 45.33))
        l09 = Bezier(l08 @ 1, (12.01, 46.07), (11.84, 46.43), (11.43, 46.69))
        l10 = Bezier(l09 @ 1, (10.98, 46.97), (10.07, 46.7), (9.47, 46.1))
        l11 = Bezier(l10 @ 1, (9.03, 45.65), (8.88, 45.31), (8.84, 44.65))
        l12 = Bezier(l11 @ 1, (8.78, 43.6), (9.11, 42.26), (9.72, 41.0))
        l13 = Bezier(l12 @ 1, (10.43, 39.54), (11.52, 38.2), (12.78, 37.22))
        l14 = Bezier(l13 @ 1, (15.36, 35.23), (16.58, 33.76), (17.45, 31.62))
        l15 = Bezier(l14 @ 1, (17.91, 30.49), (18.22, 29.27), (18.4, 27.8))
        l16 = Bezier(l15 @ 1, (18.53, 26.78), (18.52, 23.69), (18.37, 22.61))
        l17 = Bezier(l16 @ 1, (17.8, 18.23), (16.15, 14.7), (13.39, 11.94))
        l18 = Bezier(l17 @ 1, (11.89, 10.45), (10.19, 9.31), (8.09, 8.41))
        l19 = Bezier(l18 @ 1, (3.32, 6.35), (0.0, 6.64))
        mirror(about=Plane.YZ)
    make_face()

tire = revolve(Pos(Y=-wheel_diameter / 2) * tire_profile.face(), Axis.X)

with BuildSketch() as tread_pattern:
    with Locations((1, 1)):
        Trapezoid(15, 12, 60, 120, align=Align.MIN)
    with Locations((1, 8)):
        with GridLocations(0, 5, 1, 2):
            Rectangle(50, 2, mode=Mode.SUBTRACT)

# Define the surface and path that the tread pattern will be wrapped onto
half_road_surface = Face.revolve(Pos(Y=-wheel_diameter / 2) * l00, 360, Axis.X)
tread_path = half_road_surface.edges().sort_by(Axis.X)[0]

# Wrap the planar tread pattern onto the tire's outside surface
tread_faces = half_road_surface.wrap_faces(tread_pattern.faces(), tread_path)

# Mirror the faces to the other half of the tire
tread_faces.extend([mirror(t, Plane.YZ) for t in tread_faces])

# Thicken the tread to become solid nubs
```

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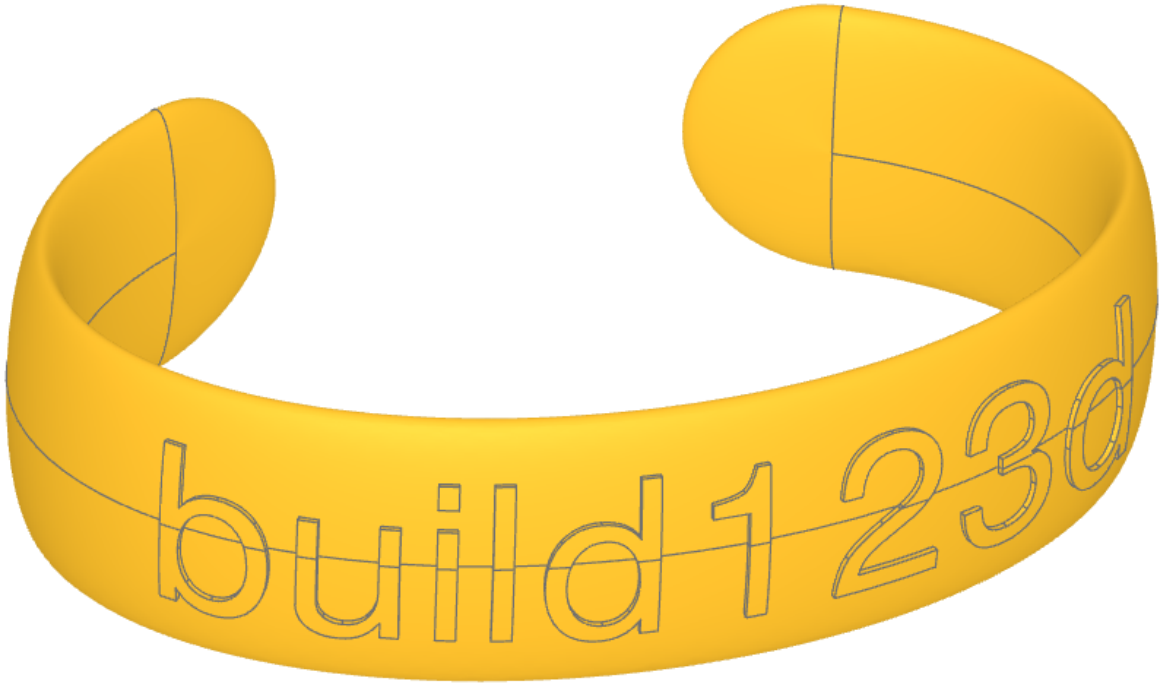
```
# tread_prime = [Solid.thicken(f, 3 * MM) for f in tread_faces]
tread_prime = [thicken(f, 3 * MM) for f in tread_faces]

# Copy the nubs around the whole tire
tread = [Rot(X=r) * copy.copy(t) for t in tread_prime for r in range(0, 360, 2)]

show(tire, tread)
```

Bracelet



Doubly-curved bracelet with an embossed label

This model is a good "stress test" for OCCT because most of the final boundary surfaces are *freeform* (not analytic planes/cylinders/spheres). The geometry is assembled from:

- a swept center section (using a curved solid end-face as the sweep profile)
- two freeform "tip caps" built as Gordon surfaces (network of curves)
- an optional embossed text label projected onto a curved solid
- alignment holes for splitting/printing/assembly

Key techniques demonstrated:

- using location_at/position_at/tangent (%) to extract local frames & tangents
- projecting curves onto a non-planar surface to create "true" 3D guide curves
- Gordon surfaces to build high-quality doubly-curved skins
- projecting faces (text) onto a complex solid and thickening them

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show

# Define input parameters
# - radii: ellipse radii (X, Y) controlling the bracelet centerline shape
# - width: bracelet width (along Z for the center sweep)
# - thickness: bracelet thickness (radial thickness of the cross section)
# - opening_angle: the missing angle that creates the wrist opening
# - label_str: optional text to emboss on the outside surface
# - Define input parameters
# radii, width, thickness, opening_angle, label_str = (45, 30), 25, 5, 80, "build123d"
radii, width, thickness, opening_angle, label_str = (45, 30), 25, 5, 80, ""

# Step 1: Create an elliptical arc defining the *centerline* of the bracelet.
# The arc is truncated to leave an opening (the "gap" where the bracelet goes on).
# Angles are in degrees; 270° points downward, which keeps the opening centered at the␣
↪bottom.
center_arc = EllipticalCenterArc(
    (0, 0), *radii, 270 + opening_angle / 2, arc_size=360 - opening_angle
)

# Step 2: Create HALF of the end cross-section, positioned at the end of the arc.
# We build only half so we can later mirror it to enforce symmetry and reduce
# curve-network complexity when building the freeform tip.
#
# location_at(1) returns a local coordinate frame at the arc end (tangent-aware).
# x_dir is chosen so the section's local "X" is well-defined and stable.
end_center_arc = center_arc.location_at(1, x_dir=(0, 0, 1))
half_x_section = EllipticalCenterArc(
    (0, 0), width / 2, thickness / 2, 90, arc_size=180
).locate(end_center_arc)

# Step 3: Create a doubly-curved "tip edge" curve.
# The tip edge must live in 3D and conform to the outside of the bracelet.
# To do that, we:
#   1) create a surface by extruding the center_arc into a sheet (a ribbon surface)
#   2) build a planar arc in a local frame at the end of that surface
#   3) project the planar arc onto the curved surface to get a true 3D curve
#
# The resulting tip_arc is a 3D edge that naturally matches the bracelet curvature.
center_surface = -Face.extrude(center_arc, (0, 0, 2 * width)).moved(
    Location((0, 0, -width), (0, 0, 180))
)
tip_center_loc = -center_surface.location_at(center_arc @ 1, x_dir=(1, 0, 0))
normal_at_tip_center = tip_center_loc.z_axis.direction

# A planar arc that would represent the outer boundary of the tip *if* the surface
# were flat. We immediately project it to make it truly conformal in 3D.
planar_tip_arc = CenterArc((0, 0), width / 2, 270, 180).locate(tip_center_loc).edge()
tip_arc = planar_tip_arc.project_to_shape(center_surface, -normal_at_tip_center)[0]
```

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```
# Step 4: Build the tip as a Gordon surface (a surface fit through a curve network).
# Gordon surfaces are ideal when:
#   - you don't have an obvious analytic surface
#   - curvature changes in two directions (doubly-curved "cap")
#   - you can define a consistent set of profile curves + guide curves
#
# Here:
#   - profiles define "across the tip" shape (section -> bulged spline -> mirrored␣
↪section)
#   - guides define "along the tip" rails (start point -> projected 3D arc -> end point)
#
# Tangents are used to encourage smoothness where the tip joins the swept center section.
profile = Spline(
    half_x_section @ 0,
    tip_arc @ 0.5,
    half_x_section @ 1,
    tangents=(center_arc % 1, -(center_arc % 1)),
)
tip_surface = Face.make_gordon_surface(
    profiles=[half_x_section, profile, half_x_section.mirror(Plane.XY)],
    guides=[half_x_section @ 0, tip_arc, half_x_section @ 1],
)

# Step 5: Close the tip surface into a watertight Solid.
# tip_surface is the outer "skin"; we create a side face from its boundary wire
# and make a shell, then a solid.
tip_side = Face(tip_surface.wire())
tip = Solid(Shell([tip_side, tip_surface]))

# Step 6: Sweep the *flat end face* of the tip around the center arc.
# This is the trick that makes the center section compatible with the freeform tip:
# the sweep profile is the same face that bounds the tip, so the join is naturally␣
↪aligned.
center_section = sweep(tip_side, center_arc).solid()

# Step 7: Assemble the bracelet from the center and two mirrored tips.
# Mirror across YZ to create the opposite end cap.
bracelet = Solid() + [tip, center_section, tip.mirror(Plane.YZ)]

# Step 8: Add an embossed label.
# This is often the hardest operation for OCCT in this model:
# projecting text onto a doubly-curved surface can create many small faces/edges,
# and thickening them adds even more boolean complexity.
if label_str:
    label = Text(label_str, font_size=width * 0.8, align=Align.CENTER)

    # Project the text onto the bracelet using a path-based placement along center_arc.
    # The parameter offsets the label so it sits centered along arc-length.
    p_labels = bracelet.project_faces(
        label, center_arc, 0.5 - 0.5 * (label.bounding_box().size.X) / center_arc.length
    )
    # Turn the projected faces into solids via thickening (embossing).
```

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```
    embossed_label = [Solid.thicken(f, 0.5) for f in p_labels.faces()]
    bracelet += embossed_label

# Step 9: Add alignment holes to aid assembly after 3D printing in two halves.
# These are placed at evenly spaced locations along the arc (including both ends).
# A small clearance (+0.15) is included for typical FDM tolerances.
alignment_holes = [
    Pos(p) * Cylinder(1.75 / 2 + 0.15, 8)
    for p in [center_arc.position_at(i / 4) for i in range(5)]
]
bracelet -= alignment_holes

show(bracelet)
```

Former build123d Logo

This example creates the former build123d logo (new logo was created in the end of 2023).

Using text and lines to create the first build123d logo. The builder mode example also generates the SVG file *logo.svg*.

Reference Implementation (Builder Mode)

```
with BuildSketch() as logo_text:
    Text("123d", font_size=10, align=(Align.MIN, Align.MIN))
    font_height = logo_text.vertices().sort_by(Axis.Y)[-1].Y

with BuildSketch() as build_text:
    Text("build", font_size=5, align=(Align.CENTER, Align.CENTER))
    build_bb = bounding_box(build_text.sketch, mode=Mode.PRIVATE)
    build_vertices = build_bb.vertices().sort_by(Axis.X)
    build_width = build_vertices[-1].X - build_vertices[0].X

with BuildLine() as one:
    l1 = Line((font_height * 0.3, 0), (font_height * 0.3, font_height))
    TangentArc(l1 @ 1, (0, font_height * 0.7), tangent=(l1 % 1) * -1)

with BuildSketch() as two:
    with Locations((font_height * 0.35, 0)):
        Text("2", font_size=10, align=(Align.MIN, Align.MIN))

with BuildPart() as three_d:
    with BuildSketch(Plane((font_height * 1.1, 0))):
        Text("3d", font_size=10, align=(Align.MIN, Align.MIN))
    extrude(amount=font_height * 0.3)
    logo_width = three_d.vertices().sort_by(Axis.X)[-1].X

with BuildLine() as arrow_left:
    t1 = TangentArc((0, 0), (1, 0.75), tangent=(1, 0))
    mirror(t1, Plane.XZ)

ext_line_length = font_height * 0.5
dim_line_length = (logo_width - build_width - 2 * font_height * 0.05) / 2
with BuildLine() as extension_lines:
    l1 = Line((0, -font_height * 0.1), (0, -ext_line_length - font_height * 0.1))
    l2 = Line(
        (logo_width, -font_height * 0.1),
        (logo_width, -ext_line_length - font_height * 0.1),
    )
    with Locations(l1 @ 0.5):
        add(arrow_left.line)
    with Locations(l2 @ 0.5):
        add(arrow_left.line, rotation=180.0)
    Line(l1 @ 0.5, l1 @ 0.5 + Vector(dim_line_length, 0))
    Line(l2 @ 0.5, l2 @ 0.5 - Vector(dim_line_length, 0))

# Precisely center the build Faces
with BuildSketch() as build:
    with Locations(
        (l1 @ 0.5 + l2 @ 0.5) / 2
        - Vector((build_vertices[-1].X + build_vertices[0].X) / 2, 0)
    ):
        add(build_text.sketch)
```

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```
if True:
    logo = Compound(
        children=[
            one.line,
            two.sketch,
            three_d.part,
            extension_lines.line,
            build.sketch,
        ]
    )

    # logo.export_step("logo.step")
    def add_svg_shape(svg: ExportSVG, shape: Shape, color: tuple[float, float, float]):
        global counter
        try:
            counter += 1
        except:
            counter = 1

        visible, _hidden = shape.project_to_viewport(
            (-5, 1, 10), viewport_up=(0, 1, 0), look_at=(0, 0, 0)
        )
        if color is not None:
            svg.add_layer(str(counter), fill_color=color, line_weight=1)
        else:
            svg.add_layer(str(counter), line_weight=1)
        svg.add_shape(visible, layer=str(counter))

    svg = ExportSVG(scale=20)
    add_svg_shape(svg, logo, None)
    # add_svg_shape(svg, Compound(children=[one.line, extension_lines.line]), None)
    # add_svg_shape(svg, Compound(children=[two.sketch, build.sketch]), (170, 204, 255))
    # add_svg_shape(svg, three_d.part, (85, 153, 255))
    svg.write("logo.svg")

show_object(one, name="one")
show_object(two, name="two")
show_object(three_d, name="three_d")
show_object(extension_lines, name="extension_lines")
show_object(build, name="build")
```

Reference Implementation (Algebra Mode)

```
logo_text = Text("123d", font_size=10, align=Align.MIN)
font_height = logo_text.vertices().sort_by(Axis.Y).last.Y

build_text = Text("build", font_size=5, align=Align.CENTER)
build_bb = build_text.bounding_box()
build_width = build_bb.max.X - build_bb.min.X
```

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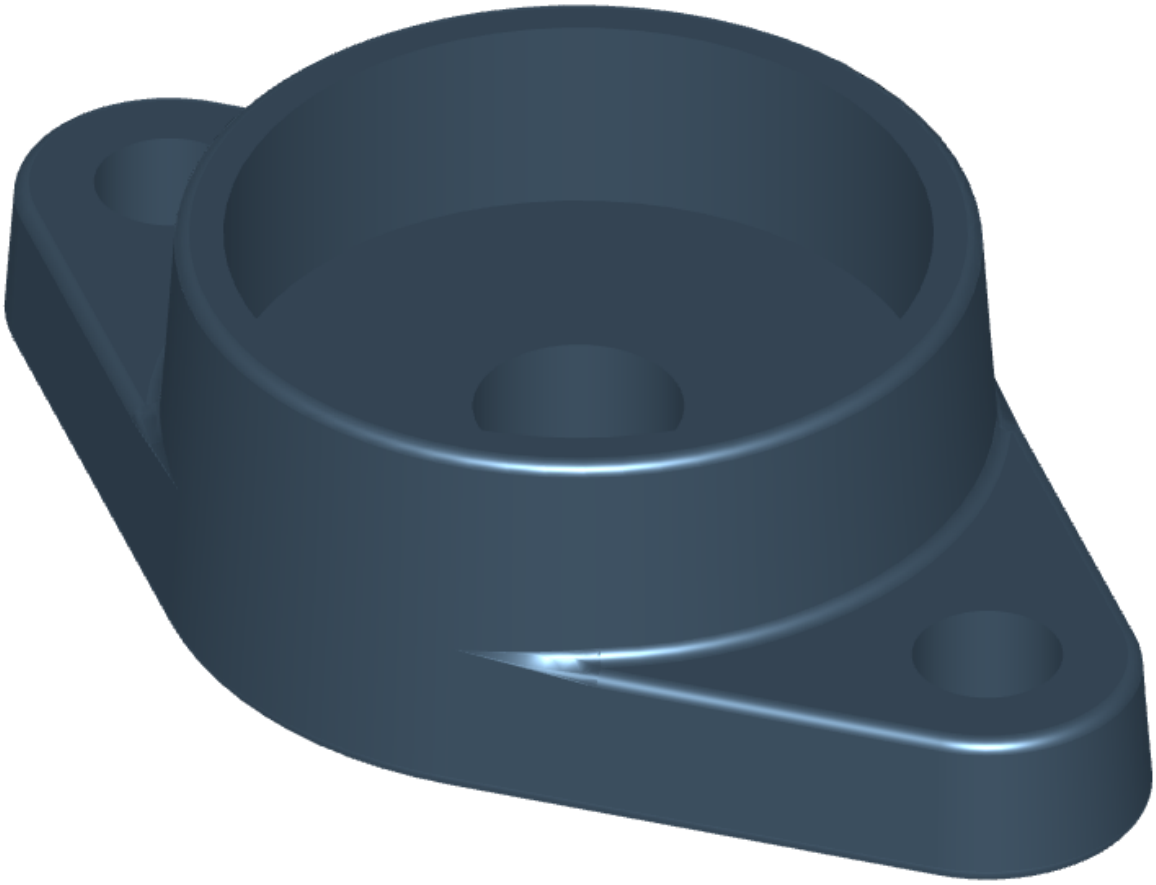
```
l1 = Line((font_height * 0.3, 0), (font_height * 0.3, font_height))
one = l1 + TangentArc(l1 @ 1, (0, font_height * 0.7), tangent=(l1 % 1) * -1)

two = Pos(font_height * 0.35, 0) * Text("2", font_size=10, align=Align.MIN)

three_d = Text("3d", font_size=10, align=Align.MIN)
three_d = Pos(font_height * 1.1, 0) * extrude(three_d, amount=font_height * 0.3)
logo_width = three_d.vertices().sort_by(Axis.X).last.X

t1 = TangentArc((0, 0), (1, 0.75), tangent=(1, 0))
arrow_left = t1 + mirror(t1, Plane.XZ)

ext_line_length = font_height * 0.5
dim_line_length = (logo_width - build_width - 2 * font_height * 0.05) / 2

l1 = Line((0, -font_height * 0.1), (0, -ext_line_length - font_height * 0.1))
l2 = Line(
    (logo_width, -font_height * 0.1),
    (logo_width, -ext_line_length - font_height * 0.1),
)
extension_lines = Curve() + (l1 + l2)
extension_lines += Pos(*(l1 @ 0.5)) * arrow_left
extension_lines += (Pos(*(l2 @ 0.5)) * Rot(Z=180)) * arrow_left
extension_lines += Line(l1 @ 0.5, l1 @ 0.5 + Vector(dim_line_length, 0))
extension_lines += Line(l2 @ 0.5, l2 @ 0.5 - Vector(dim_line_length, 0))

# Precisely center the build Faces
p1 = Pos((l1 @ 0.5 + l2 @ 0.5) / 2 - Vector((build_bb.max.X + build_bb.min.X) / 2, 0))
build = p1 * build_text

cmpd = Compound([three_d, two, one, build, extension_lines])

show_object(cmpd, name="compound")
```

Cast Bearing Unit



This example demonstrates the creation of a castable flanged bearing housing using the *draft* operation to add appropriate draft angles for mold release.

Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show

A, A1, Db2, H, J = 26, 11, 57, 98.5, 76.5
with BuildPart() as oval_flanged_bearing_unit:
    with BuildSketch() as plan:
        housing = Circle(Db2 / 2)
        with GridLocations(J, 0, 2, 1) as bolt_centers:
            Circle((H - J) / 2)
        make_hull()
    extrude(amount=A1)
    extrude(housing, amount=A)
    drafted_faces = oval_flanged_bearing_unit.faces().filter_by(Axis.Z, reverse=True)
    draft(drafted_faces, Plane.XY, 4)
    fillet(oval_flanged_bearing_unit.edges(), 1)
    with Locations(oval_flanged_bearing_unit.faces().sort_by(Axis.Z)[-1]):
```

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```
        CounterBoreHole(14 / 2, 47 / 2, 14)
    with Locations(*bolt_centers):
        Hole(5)

oval_flanged_bearing_unit.part.color = Color(0x4C6377)

show(oval_flanged_bearing_unit)
```

Canadian Flag Blowing in The Wind

A Canadian Flag blowing in the wind created by projecting planar faces onto a non-planar face (the_wind).

This example also demonstrates building complex lines that snap to existing features.

More Images

Reference Implementation (Builder Mode)

```
def surface(amplitude, u, v):
    """Calculate the surface displacement of the flag at a given position"""
    return v * amplitude / 20 * cos(3.5 * pi * u) + amplitude / 10 * v * sin(
        1.1 * pi * v
    )
```

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```
# Note that the surface to project on must be a little larger than the faces
# being projected onto it to create valid projected faces
the_wind = Face.make_surface_from_array_of_points(
    [
        [
            Vector(
                width * (v * 1.1 / 40 - 0.05),
                height * (u * 1.2 / 40 - 0.1),
                height * surface(wave_amplitude, u / 40, v / 40) / 2,
            )
            for u in range(41)
        ]
        for v in range(41)
    ]
)
with BuildSketch(Plane.XY.offset(10)) as west_field_builder:
    Rectangle(width / 4, height, align=(Align.MIN, Align.MIN))
west_field_planar = west_field_builder.sketch.faces()[0]
east_field_planar = west_field_planar.mirror(Plane.YZ.offset(width / 2))

with BuildSketch(Plane((width / 2, 0, 10))) as center_field_builder:
    Rectangle(width / 2, height, align=(Align.CENTER, Align.MIN))
    with BuildSketch(
        Plane((width / 2, 0, 10)), mode=Mode.SUBTRACT
    ) as maple_leaf_builder:
        with BuildLine() as outline:
            l1 = Polyline((0.0000, 0.0771), (0.0187, 0.0771), (0.0094, 0.2569))
            l2 = Polyline((0.0325, 0.2773), (0.2115, 0.2458), (0.1873, 0.3125))
            RadiusArc(l1 @ 1, l2 @ 0, 0.0271)
            l3 = Polyline((0.1915, 0.3277), (0.3875, 0.4865), (0.3433, 0.5071))
            TangentArc(l2 @ 1, l3 @ 0, tangent=l2 % 1)
            l4 = Polyline((0.3362, 0.5235), (0.375, 0.6427), (0.2621, 0.6188))
            SagittaArc(l3 @ 1, l4 @ 0, 0.003)
            l5 = Polyline((0.2469, 0.6267), (0.225, 0.6781), (0.1369, 0.5835))
            ThreePointArc(
                l4 @ 1, (l4 @ 1 + l5 @ 0) * 0.5 + Vector(-0.002, -0.002), l5 @ 0
            )
            l6 = Polyline((0.1138, 0.5954), (0.1562, 0.8146), (0.0881, 0.7752))
            Spline(
                l5 @ 1,
                l6 @ 0,
                tangents=(l5 % 1, l6 % 0),
                tangent_scalars=(2, 2),
            )
            l7 = Line((0.0692, 0.7808), (0.0000, 0.9167))
            TangentArc(l6 @ 1, l7 @ 0, tangent=l6 % 1)
            mirror(outline.edges(), Plane.YZ)
        make_face()
        scale(by=height)
maple_leaf_planar = maple_leaf_builder.sketch.faces()[0]
```

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```
center_field_planar = center_field_builder.sketch.faces()[0]

west_field = west_field_planar.project_to_shape(the_wind, (0, 0, -1))[0]
west_field.color = Color("red")
east_field = east_field_planar.project_to_shape(the_wind, (0, 0, -1))[0]
east_field.color = Color("red")
center_field = center_field_planar.project_to_shape(the_wind, (0, 0, -1))[0]
center_field.color = Color("white")
maple_leaf = maple_leaf_planar.project_to_shape(the_wind, (0, 0, -1))[0]
maple_leaf.color = Color("red")

canadian_flag = Compound(children=[west_field, east_field, center_field, maple_leaf])
show(Rot(90, 0, 0) * canadian_flag)
```

Reference Implementation (Algebra Mode)

```
def surface(amplitude, u, v):
    """Calculate the surface displacement of the flag at a given position"""
    return v * amplitude / 20 * cos(3.5 * pi * u) + amplitude / 10 * v * sin(
        1.1 * pi * v
    )


# Note that the surface to project on must be a little larger than the faces
# being projected onto it to create valid projected faces
the_wind = Face.make_surface_from_array_of_points(
    [
        [
            Vector(
                width * (v * 1.1 / 40 - 0.05),
                height * (u * 1.2 / 40 - 0.1),
                height * surface(wave_amplitude, u / 40, v / 40) / 2,
            )
            for u in range(41)
        ]
        for v in range(41)
    ]
)

field_planar = Plane.XY.offset(10) * Rectangle(width / 4, height, align=Align.MIN)
west_field_planar = field_planar.faces()[0]
east_field_planar = mirror(west_field_planar, Plane.YZ.offset(width / 2))

l1 = Polyline((0.0000, 0.0771), (0.0187, 0.0771), (0.0094, 0.2569))
l2 = Polyline((0.0325, 0.2773), (0.2115, 0.2458), (0.1873, 0.3125))
r1 = RadiusArc(l1 @ 1, l2 @ 0, 0.0271)
l3 = Polyline((0.1915, 0.3277), (0.3875, 0.4865), (0.3433, 0.5071))
r2 = TangentArc(l2 @ 1, l3 @ 0, tangent=l2 % 1)
l4 = Polyline((0.3362, 0.5235), (0.375, 0.6427), (0.2621, 0.6188))
r3 = SagittaArc(l3 @ 1, l4 @ 0, 0.003)
l5 = Polyline((0.2469, 0.6267), (0.225, 0.6781), (0.1369, 0.5835))
```

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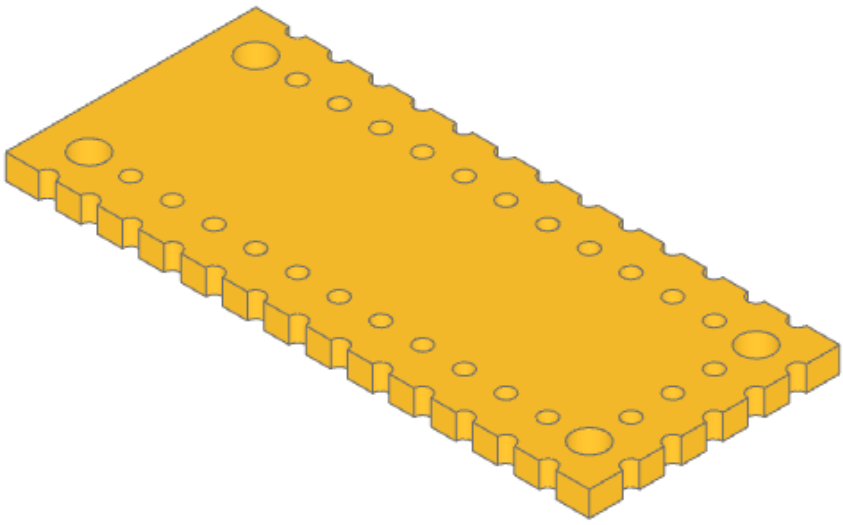
```
r4 = ThreePointArc(l4 @ 1, (l4 @ 1 + l5 @ 0) * 0.5 + Vector(-0.002, -0.002), l5 @ 0)
l6 = Polyline((0.1138, 0.5954), (0.1562, 0.8146), (0.0881, 0.7752))
s = Spline(
    l5 @ 1,
    l6 @ 0,
    tangents=(l5 % 1, l6 % 0),
    tangent_scalars=(2, 2),
)
l7 = Line((0.0692, 0.7808), (0.0000, 0.9167))
r5 = TangentArc(l6 @ 1, l7 @ 0, tangent=l6 % 1)

outline = l1 + [l2, r1, l3, r2, l4, r3, l5, r4, l6, s, l7, r5]
outline += mirror(outline, Plane.YZ)

maple_leaf_planar = make_face(outline)

center_field_planar = (
    Rectangle(1, 1, align=(Align.CENTER, Align.MIN)) - maple_leaf_planar
)


def scale_move(obj):
    return Plane((width / 2, 0, 10)) * scale(obj, height)


def project(obj):
    return obj.faces()[0].project_to_shape(the_wind, (0, 0, -1))[0]


maple_leaf_planar = scale_move(maple_leaf_planar)
center_field_planar = scale_move(center_field_planar)

west_field = project(west_field_planar)
west_field.color = Color("red")
east_field = project(east_field_planar)
east_field.color = Color("red")
center_field = project(center_field_planar)
center_field.color = Color("white")
maple_leaf = project(maple_leaf_planar)
maple_leaf.color = Color("red")

canadian_flag = Compound(children=[west_field, east_field, center_field, maple_leaf])
show(Rot(90, 0, 0) * canadian_flag)
```

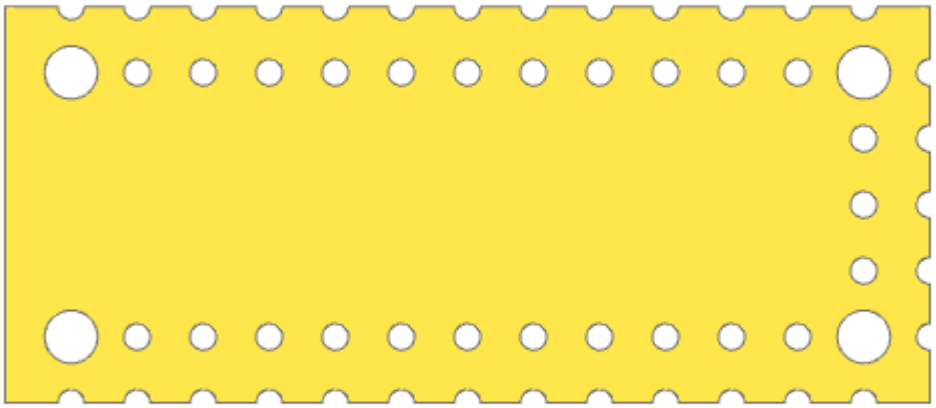
Circuit Board With Holes



This example demonstrates placing holes around a part.

- Builder mode uses *Locations* context to place the positions.
- Algebra mode uses *product* and *range* to calculate the positions.

More Images



Reference Implementation (Builder Mode)

```
with BuildPart() as pcb:
    with BuildSketch():
        Rectangle(pcb_length, pcb_width)

        for i in range(65 // 5):
            x = i * 5 - 30
            with Locations((x, -15), (x, -10), (x, 10), (x, 15)):
                Circle(1, mode=Mode.SUBTRACT)
        for i in range(30 // 5 - 1):
            y = i * 5 - 10
            with Locations((30, y), (35, y)):
                Circle(1, mode=Mode.SUBTRACT)
        with GridLocations(60, 20, 2, 2):
            Circle(2, mode=Mode.SUBTRACT)
    extrude(amount=pcb_height)

show_object(pcb.part.wrapped)
```

Reference Implementation (Algebra Mode)

```
x_coords = product(range(65 // 5), (-15, -10, 10, 15))
y_coords = product((30, 35), range(30 // 5 - 1))

pcb = Rectangle(pcb_length, pcb_width)
pcb -= [Pos(i * 5 - 30, y) * Circle(1) for i, y in x_coords]
```

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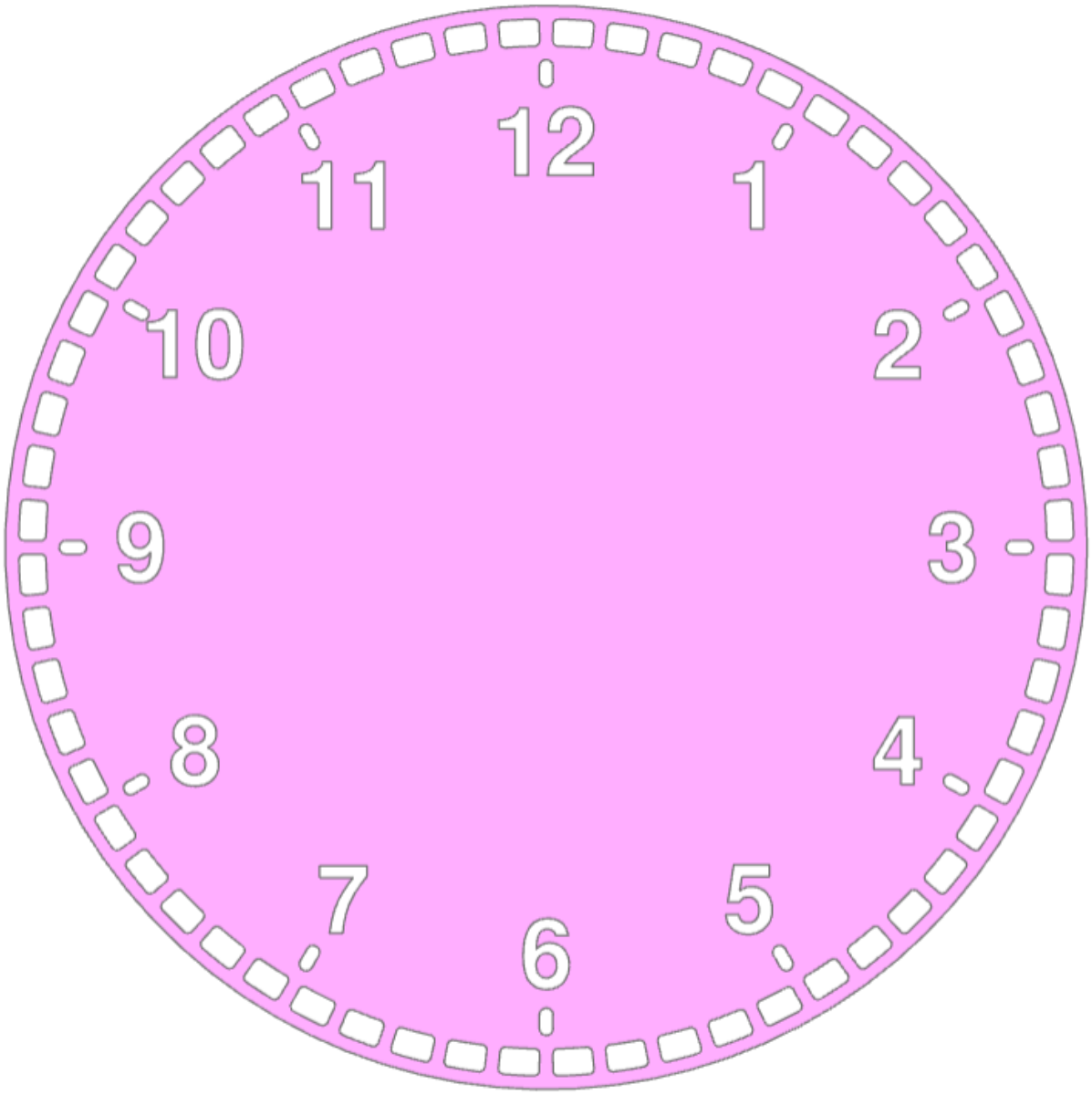
```
pcb -= [Pos(x, i * 5 - 10) * Circle(1) for x, i in y_coords]
pcb -= [loc * Circle(2) for loc in GridLocations(60, 20, 2, 2)]

pcb = extrude(pcb, pcb_height)

show(pcb)
```

Clock Face



Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show

clock_radius = 10
with BuildSketch() as minute_indicator:
    with BuildLine() as outline:
        l1 = CenterArc((0, 0), clock_radius * 0.975, 0.75, 4.5)
        l2 = CenterArc((0, 0), clock_radius * 0.925, 0.75, 4.5)
        Line(l1 @ 0, l2 @ 0)
        Line(l1 @ 1, l2 @ 1)
    make_face()
    fillet(minute_indicator.vertices(), radius=clock_radius * 0.01)

with BuildSketch() as clock_face:
    Circle(clock_radius)
    with PolarLocations(0, 60):
        add(minute_indicator.sketch, mode=Mode.SUBTRACT)
    with PolarLocations(clock_radius * 0.875, 12):
        SlotOverall(clock_radius * 0.05, clock_radius * 0.025, mode=Mode.SUBTRACT)
    for hour in range(1, 13):
        with PolarLocations(clock_radius * 0.75, 1, -hour * 30 + 90, 360, rotate=False):
            Text(
                str(hour),
                font_size=clock_radius * 0.175,
                font_style=FontStyle.BOLD,
                mode=Mode.SUBTRACT,
            )

show(clock_face)
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show

clock_radius = 10

l1 = CenterArc((0, 0), clock_radius * 0.975, 0.75, 4.5)
l2 = CenterArc((0, 0), clock_radius * 0.925, 0.75, 4.5)
l3 = Line(l1 @ 0, l2 @ 0)
l4 = Line(l1 @ 1, l2 @ 1)
minute_indicator = make_face([l1, l3, l2, l4])
minute_indicator = fillet(minute_indicator.vertices(), radius=clock_radius * 0.01)

clock_face = Circle(clock_radius)
clock_face -= PolarLocations(0, 60) * minute_indicator
clock_face -= PolarLocations(clock_radius * 0.875, 12) * SlotOverall(
    clock_radius * 0.05, clock_radius * 0.025
)

clock_face -= [
```

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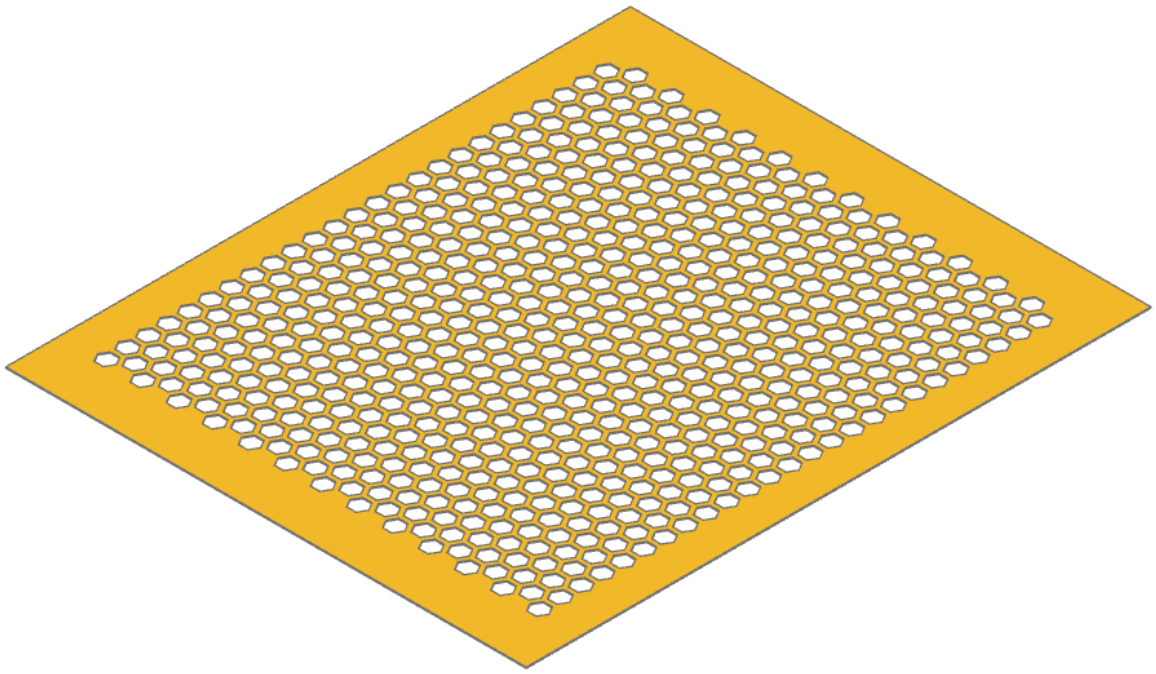
```
    loc
    * Text(
        str(hour + 1),
        font_size=clock_radius * 0.175,
        font_style=FontStyle.BOLD,
        align=Align.CENTER,
    )
    for hour, loc in enumerate(
        PolarLocations(clock_radius * 0.75, 12, 60, -360, rotate=False)
    )
]

show(clock_face)
```

The Python code utilizes the build123d library to create a 3D model of a clock face. It defines a minute indicator with arcs and lines, applying fillets, and then integrates it into the clock face sketch. The clock face includes a circular outline, hour labels, and slots at specified positions. The resulting 3D model represents a detailed and visually appealing clock design.

PolarLocations are used to position features on the clock face.

Fast Grid Holes



Reference Implementation (Algebra Mode)

```
import timeit
from build123d import *
from ocp_vscode import show
```

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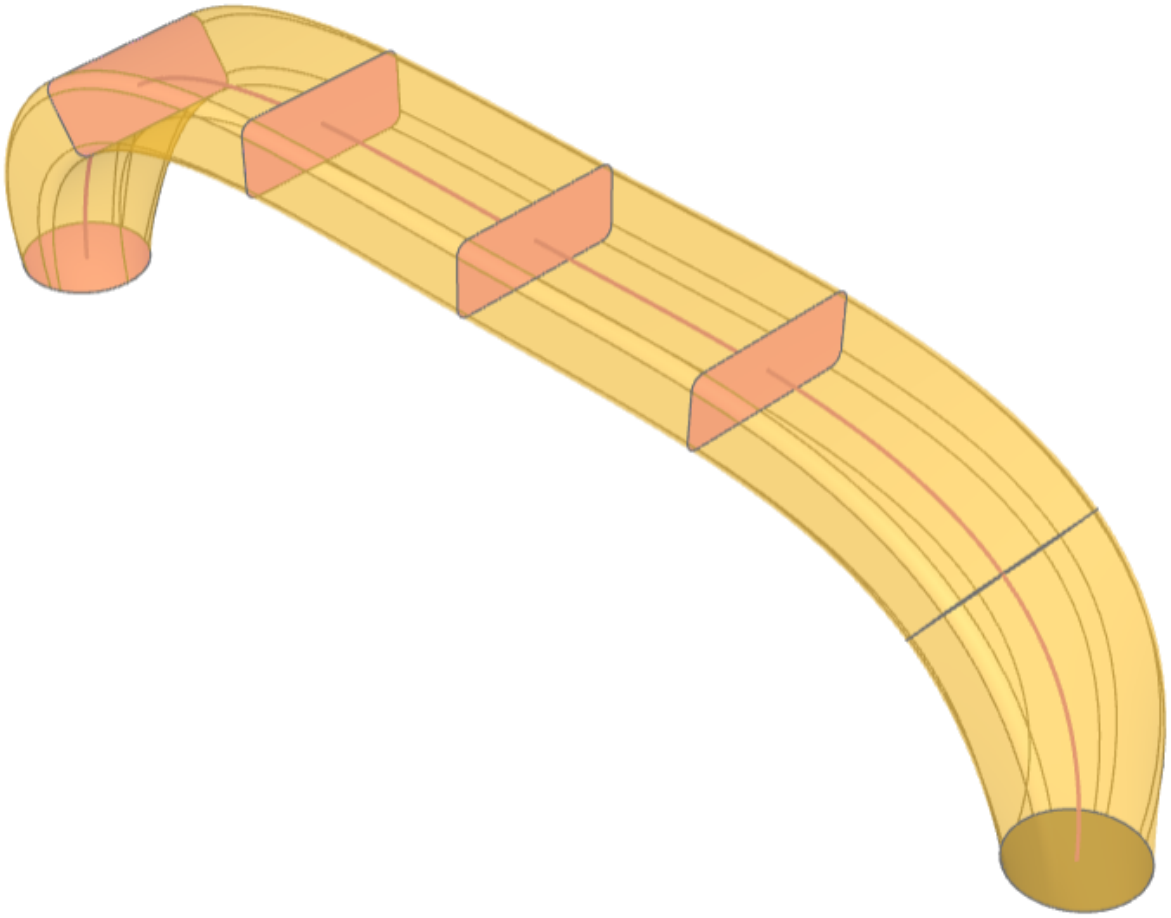
```
start_time = timeit.default_timer()

# Calculate the locations of 625 holes
major_r = 10
hole_locs = HexLocations(major_r, 25, 25)

# Create wires for both the perimeter and all the holes
face_perimeter = Rectangle(500, 600).wire()
hex_hole = RegularPolygon(major_r - 1, 6, major_radius=True).wire()
holes = hole_locs * hex_hole

# Create a new Face from the perimeter and hole wires
grid_pattern = Face(face_perimeter, holes)

# Extrude to a 3D part
grid = extrude(grid_pattern, 1)

print(f"Time: {timeit.default_timer() - start_time:0.3f}s")
show(grid)
```

This example demonstrates an efficient approach to creating a large number of holes (625 in this case) in a planar part using build123d.

Instead of modeling and subtracting 3D solids for each hole—which is computationally expensive—this method constructs a 2D Face from an outer perimeter wire and a list of hole wires. The entire face is then extruded in a single operation to form the final 3D object. This approach significantly reduces modeling time and complexity.

The hexagonal hole pattern is generated using HexLocations, and each location is populated with a hexagonal wire. These wires are passed directly to the Face constructor as holes. On a typical Linux laptop, this script completes in approximately 1.02 seconds, compared to substantially longer runtimes for boolean subtraction of individual holes in 3D.

Handle



Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show_object

segment_count = 6

with BuildPart() as handle:
    # Create a path for the sweep along the handle - added to pending_edges
    with BuildLine() as handle_center_line:
        Spline(
            (-10, 0, 0),
            (0, 0, 5),
            (10, 0, 0),
            tangents=((0, 0, 1), (0, 0, -1)),
            tangent_scalars=(1.5, 1.5),
        )

    # Create the cross sections - added to pending_faces
```

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```
    for i in range(segment_count + 1):
        with BuildSketch(handle_center_line.line ^ (i / segment_count)) as section:
            if i % segment_count == 0:
                Circle(1)
            else:
                Rectangle(1.25, 3)
                fillet(section.vertices(), radius=0.2)
    # Record the sections for display
    sections = handle.pending_faces

    # Create the handle by sweeping along the path
    sweep(multisection=True)

assert abs(handle.part.volume - 94.77361455046953) < 1e-3

show_object(handle_center_line.line, name="handle_center_line")
for i, section in enumerate(sections):
    show_object(section, name="section" + str(i))
show_object(handle.part, name="handle", options=dict(alpha=0.6))
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show_object

segment_count = 6

# Create a path for the sweep along the handle - added to pending_edges
handle_center_line = Spline(
    (-10, 0, 0),
    (0, 0, 5),
    (10, 0, 0),
    tangents=((0, 0, 1), (0, 0, -1)),
    tangent_scalars=(1.5, 1.5),
)

# Create the cross sections - added to pending_faces
sections = Sketch()
for i in range(segment_count + 1):
    location = handle_center_line ^ (i / segment_count)
    if i % segment_count == 0:
        circle = location * Circle(1)
    else:
        circle = location * Rectangle(1.25, 3)
        circle = fillet(circle.vertices(), radius=0.2)
    sections += circle

# Create the handle by sweeping along the path
handle = sweep(sections, path=handle_center_line, multisection=True)

show_object(handle_center_line, name="handle_path")
```

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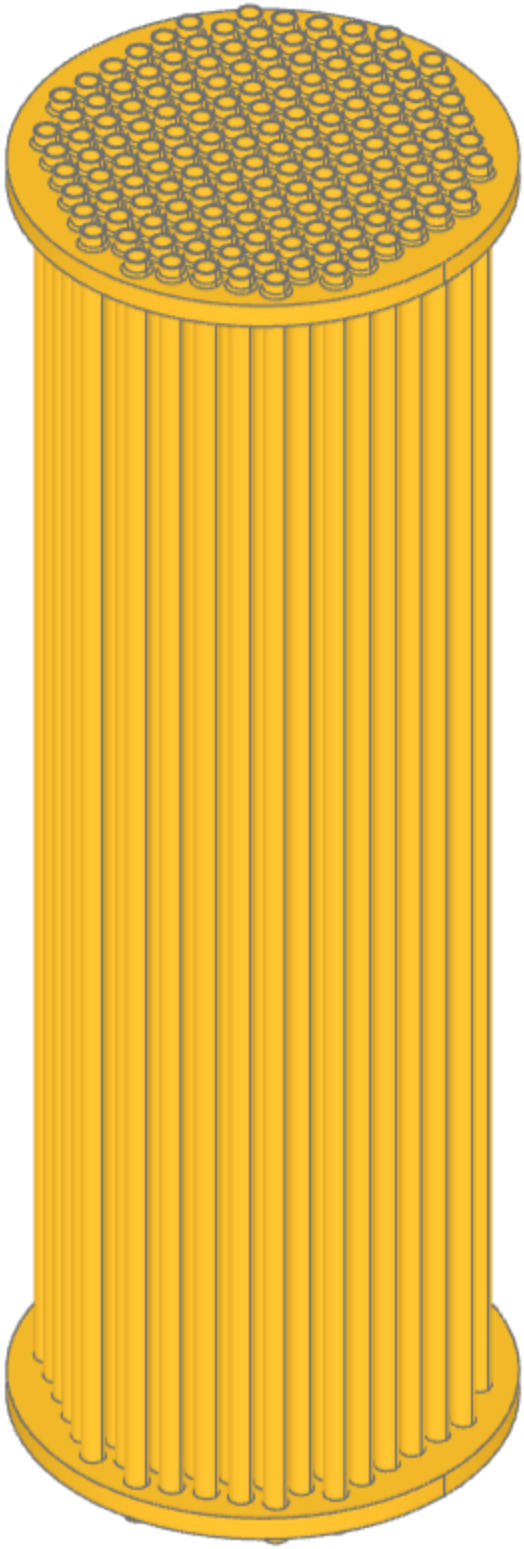
```
for i, circle in enumerate(sections):
    show_object(circle, name="section" + str(i))
show_object(handle, name="handle", options=dict(alpha=0.6))
```

This example demonstrates multisection sweep creating a drawer handle.

Heat Exchanger



Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show

exchanger_diameter = 10 * CM
exchanger_length = 30 * CM
plate_thickness = 5 * MM
# 149 tubes
tube_diameter = 5 * MM
tube_spacing = 2 * MM
tube_wall_thickness = 0.5 * MM
tube_extension = 3 * MM
bundle_diameter = exchanger_diameter - 2 * tube_diameter
fillet_radius = tube_spacing / 3
assert tube_extension > fillet_radius

# Build the heat exchanger
with BuildPart() as heat_exchanger:
    # Generate list of tube locations
    tube_locations = [
        l
        for l in HexLocations(
            radius=(tube_diameter + tube_spacing) / 2,
            x_count=exchanger_diameter // tube_diameter,
            y_count=exchanger_diameter // tube_diameter,
        )
        if l.position.length < bundle_diameter / 2
    ]
    tube_count = len(tube_locations)
    with BuildSketch() as tube_plan:
        with Locations(*tube_locations):
            Circle(radius=tube_diameter / 2)
            Circle(radius=tube_diameter / 2 - tube_wall_thickness, mode=Mode.SUBTRACT)
    extrude(amount=exchanger_length / 2)
    with BuildSketch(
        Plane(
            origin=(0, 0, exchanger_length / 2 - tube_extension - plate_thickness),
            z_dir=(0, 0, 1),
        )
    ) as plate_plan:
        Circle(radius=exchanger_diameter / 2)
        with Locations(*tube_locations):
            Circle(radius=tube_diameter / 2 - tube_wall_thickness, mode=Mode.SUBTRACT)
    extrude(amount=plate_thickness)
    half_volume_before_fillet = heat_exchanger.part.volume
    # Simulate welded tubes by adding a fillet to the outside radius of the tubes
    fillet(
        heat_exchanger.edges()
        .filter_by(GeomType.CIRCLE)
        .sort_by(SortBy.RADIUS)
        .sort_by(Axis.Z, reverse=True)[2 * tube_count : 3 * tube_count],
        radius=fillet_radius,
```

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```
    )
    half_volume_after_fillet = heat_exchanger.part.volume
    mirror(about=Plane.XY)

fillet_volume = 2 * (half_volume_after_fillet - half_volume_before_fillet)
assert abs(fillet_volume - 469.88331045553787) < 1e-3

show(heat_exchanger)
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show

exchanger_diameter = 10 * CM
exchanger_length = 30 * CM
plate_thickness = 5 * MM
# 149 tubes
tube_diameter = 5 * MM
tube_spacing = 2 * MM
tube_wall_thickness = 0.5 * MM
tube_extension = 3 * MM
bundle_diameter = exchanger_diameter - 2 * tube_diameter
fillet_radius = tube_spacing / 3
assert tube_extension > fillet_radius

# Build the heat exchanger
tube_locations = [
    l
    for l in HexLocations(
        radius=(tube_diameter + tube_spacing) / 2,
        x_count=exchanger_diameter // tube_diameter,
        y_count=exchanger_diameter // tube_diameter,
    )
    if l.position.length < bundle_diameter / 2
]

ring = Circle(tube_diameter / 2) - Circle(tube_diameter / 2 - tube_wall_thickness)
tube_plan = Sketch() + tube_locations * ring

heat_exchanger = extrude(tube_plan, exchanger_length / 2)

plate_plane = Plane(
    origin=(0, 0, exchanger_length / 2 - tube_extension - plate_thickness),
    z_dir=(0, 0, 1),
)
plate = Circle(radius=exchanger_diameter / 2) - tube_locations * Circle(
    radius=tube_diameter / 2 - tube_wall_thickness
)

heat_exchanger += extrude(plate_plane * plate, plate_thickness)
```

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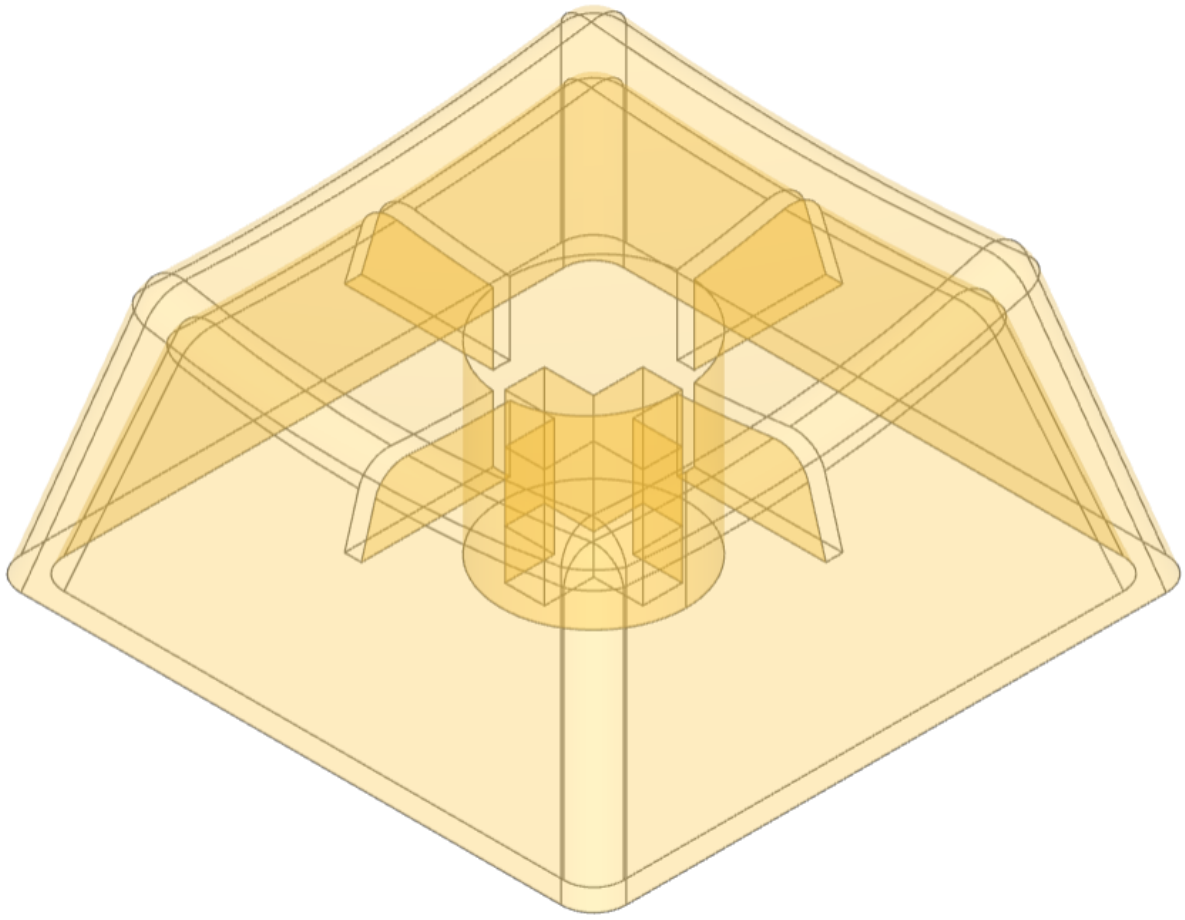
```
edges = (
    heat_exchanger.edges()
    .filter_by(GeomType.CIRCLE)
    .group_by(SortBy.RADIUS)[1]
    .group_by()[2]
)
half_volume_before_fillet = heat_exchanger.volume
heat_exchanger = fillet(edges, radius=fillet_radius)
half_volume_after_fillet = heat_exchanger.volume
heat_exchanger += mirror(heat_exchanger, Plane.XY)

fillet_volume = 2 * (half_volume_after_fillet - half_volume_before_fillet)
assert abs(fillet_volume - 469.88331045553787) < 1e-3

show(heat_exchanger)
```

This example creates a model of a parametric heat exchanger core. The positions of the tubes are defined with *HexLocations* and further limited to fit within the circular end caps. The ends of the tubes are filleted to the end plates to simulate welding.

Key Cap



Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import *

with BuildPart() as key_cap:
    # Start with the plan of the key cap and extrude it
    with BuildSketch() as plan:
        Rectangle(18 * MM, 18 * MM)
    extrude(amount=10 * MM, taper=15)
    # Create a dished top
    with Locations((0, -3 * MM, 47 * MM)):
        Sphere(40 * MM, mode=Mode.SUBTRACT, rotation=(90, 0, 0))
    # Fillet all the edges except the bottom
    fillet(
        key_cap.edges().filter_by_position(Axis.Z, 0, 30 * MM, inclusive=(False, True)),
        radius=1 * MM,
    )
    # Hollow out the key by subtracting a scaled version
    scale(by=(0.925, 0.925, 0.85), mode=Mode.SUBTRACT)

    # Add supporting ribs while leaving room for switch activation
    with BuildSketch(Plane(origin=(0, 0, 4 * MM))):
        Rectangle(15 * MM, 0.5 * MM)
        Rectangle(0.5 * MM, 15 * MM)
        Circle(radius=5.5 * MM / 2)
    # Extrude the mount and ribs to the key cap underside
    extrude(until=Until.NEXT)
    # Find the face on the bottom of the ribs to build onto
    rib_bottom = key_cap.faces().filter_by_position(Axis.Z, 4 * MM, 4 * MM)[0]
    # Add the switch socket
    with BuildSketch(rib_bottom) as cruciform:
        Circle(radius=5.5 * MM / 2)
        Rectangle(4.1 * MM, 1.17 * MM, mode=Mode.SUBTRACT)
        Rectangle(1.17 * MM, 4.1 * MM, mode=Mode.SUBTRACT)
    extrude(amount=3.5 * MM, mode=Mode.ADD)

assert abs(key_cap.part.volume - 644.8900473617498) < 1e-3

show(key_cap, alphas=[0.3])
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import *

# Taper Extrude and Extrude to "next" while creating a Cherry MX key cap
# See: https://www.cherrymx.de/en/dev.html

plan = Rectangle(18 * MM, 18 * MM)
key_cap = extrude(plan, amount=10 * MM, taper=15)

# Create a dished top
```

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```
key_cap -= Location((0, -3 * MM, 47 * MM), (90, 0, 0)) * Sphere(40 * MM)

# Fillet all the edges except the bottom
key_cap = fillet(
    key_cap.edges().filter_by_position(Axis.Z, 0, 30 * MM, inclusive=(False, True)),
    radius=1 * MM,
)

# Hollow out the key by subtracting a scaled version
key_cap -= scale(key_cap, (0.925, 0.925, 0.85))


# Add supporting ribs while leaving room for switch activation
ribs = Rectangle(17.5 * MM, 0.5 * MM)
ribs += Rectangle(0.5 * MM, 17.5 * MM)
ribs += Circle(radius=5.51 * MM / 2)

# Extrude the mount and ribs to the key cap underside
key_cap += extrude(Pos(0, 0, 4 * MM) * ribs, until=Until.NEXT, target=key_cap)


# Find the face on the bottom of the ribs to build onto
rib_bottom = key_cap.faces().filter_by_position(Axis.Z, 4 * MM, 4 * MM)[0]

# Add the switch socket
socket = Circle(radius=5.5 * MM / 2)
socket -= Rectangle(4.1 * MM, 1.17 * MM)
socket -= Rectangle(1.17 * MM, 4.1 * MM)
key_cap += extrude(Plane(rib_bottom) * socket, amount=3.5 * MM)

show(key_cap, alphas=[0.3])
```

This example demonstrates the design of a Cherry MX key cap by using extrude with a taper and extrude until next.

Maker Coin

This example creates the maker coin as defined by Angus on the Maker's Muse YouTube channel. There are two key features:

1. the use of *DoubleTangentArc* to create a smooth transition from the central dish to the outside arc, and
2. embossing the text into the top of the coin not just as a simple extrude but from a projection which results in text with even depth.

Reference Implementation (Builder Mode)

```
# Coin Parameters
diameter, thickness = 50 * MM, 10 * MM

with BuildPart() as maker_coin:
    # On XZ plane draw the profile of half the coin
    with BuildSketch(Plane.XZ) as profile:
        with BuildLine() as outline:
            l1 = Polyline((0, thickness * 0.6), (0, 0), ((diameter - thickness) / 2, 0))
            l2 = JernArc(
                start=l1 @ 1, tangent=l1 % 1, radius=thickness / 2, arc_size=300
```

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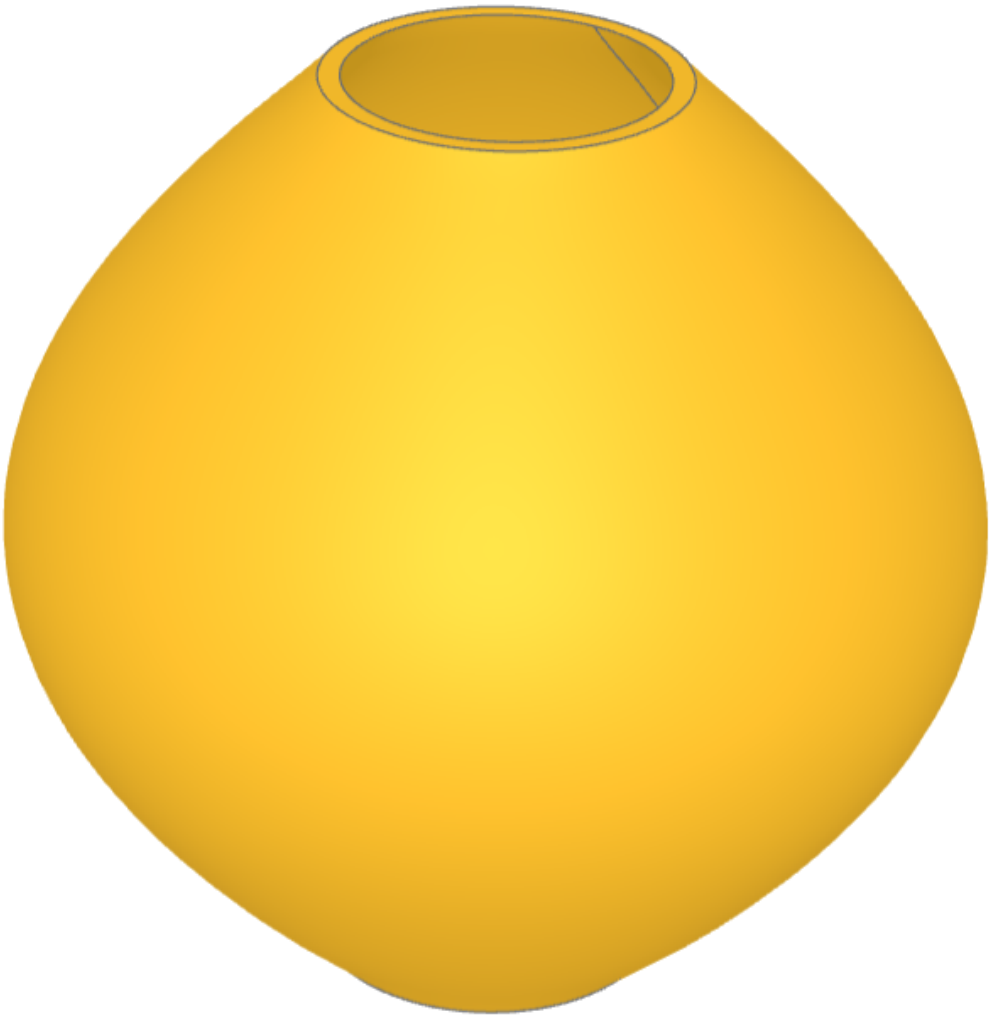
```
            )  # extend the arc beyond the intersection but not closed
            l3 = DoubleTangentArc(l1 @ 0, tangent=(1, 0), other=l2)
        make_face()  # make it a 2D shape
    revolve()  # revolve 360°

    # Pattern the detents around the coin
    with BuildSketch() as detents:
        with PolarLocations(radius=(diameter + 5) / 2, count=8):
            Circle(thickness * 1.4 / 2)
    extrude(amount=thickness, mode=Mode.SUBTRACT)  # cut away the detents

    fillet(maker_coin.edges(Select.NEW), 2)  # fillet the cut edges

    # Add an embossed label
    with BuildSketch(Plane.XY.offset(thickness)) as label:  # above coin
        Text("OS", font_size=15)
    project()  # label on top of coin
    extrude(amount=-thickness / 5, mode=Mode.SUBTRACT)  # emboss label

show(maker_coin)
```

Multi-Sketch Loft



This example demonstrates lofting a set of sketches, selecting the top and bottom by type, and shelling.

Reference Implementation (Builder Mode)

```
from math import pi, sin
from build123d import *
from ocp_vscode import show

with BuildPart() as art:
    slice_count = 10
    for i in range(slice_count + 1):
```

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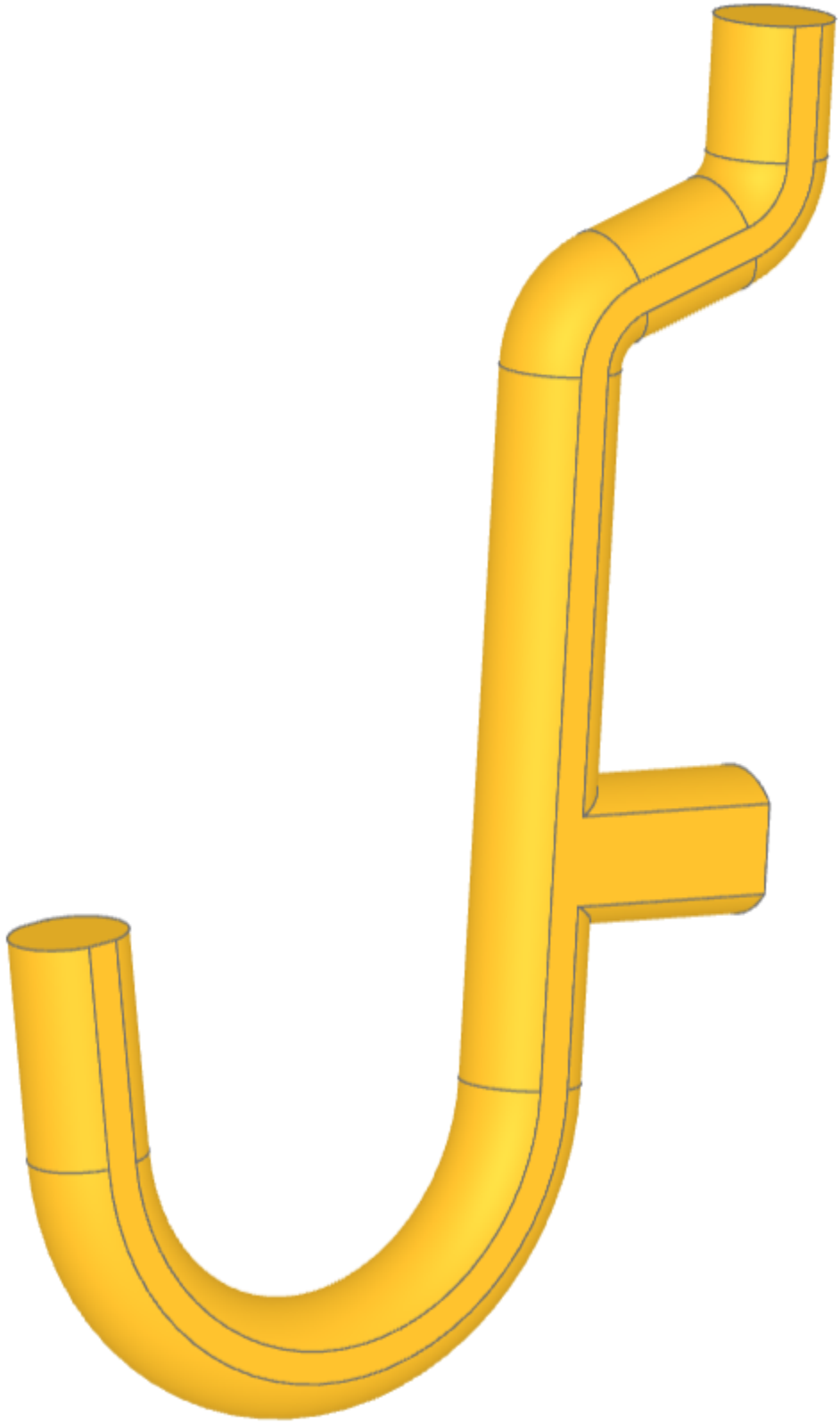
```
        with BuildSketch(Plane(origin=(0, 0, i * 3), z_dir=(0, 0, 1))) as slice:
            Circle(10 * sin(i * pi / slice_count) + 5)
    loft()
    top_bottom = art.faces().filter_by(GeomType.PLANE)
    offset(openings=top_bottom, amount=0.5)

want = 1306.3405290344635
got = art.part.volume
delta = abs(got - want)
tolerance = want * 1e-5
assert delta < tolerance, f"{delta=} is greater than {tolerance=}; {got=}, {want=}"

show(art, names=["art"])
```

Reference Implementation (Algebra Mode)

```
from math import pi, sin
from build123d import *
from ocp_vscode import show

slice_count = 10

art = Sketch()
for i in range(slice_count + 1):
    plane = Plane(origin=(0, 0, i * 3), z_dir=(0, 0, 1))
    art += plane * Circle(10 * sin(i * pi / slice_count) + 5)

art = loft(art)
top_bottom = art.faces().filter_by(GeomType.PLANE)
art = offset(art, openings=top_bottom, amount=0.5)

show(art, names=["art"])
```

Peg Board Hook



This script creates a a J-shaped pegboard hook. These hooks are commonly used for organizing tools in garages, workshops, or other spaces where tools and equipment need to be stored neatly and accessibly. The hook is created by defining a complex path and then sweeping it to define the hook. The sides of the hook are flattened to aid 3D printing.

Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show

pegd = 6.35 + 0.1  # mm ~0.25inch
c2c = 25.4  # mm 1.0inch
arcd = 7.2
both = 10
topx = 6
midx = 8
maind = 0.82 * pegd
midd = 1.0 * pegd
hookd = 23
hookx = 10
splitz = maind / 2 - 0.1
topangs = 70

with BuildPart() as mainp:
    with BuildLine(mode=Mode.PRIVATE) as sprof:
        l1 = Line((-both, 0), (c2c - arcd / 2 - 0.5, 0))
        l2 = JernArc(start=l1 @ 1, tangent=l1 % 1, radius=arcd / 2, arc_size=topangs)
        l3 = PolarLine(
            start=l2 @ 1,
            length=topx,
            direction=l2 % 1,
        )
        l4 = JernArc(start=l3 @ 1, tangent=l3 % 1, radius=arcd / 2, arc_size=-topangs)
        l5 = PolarLine(
            start=l4 @ 1,
            length=topx,
            direction=l4 % 1,
        )
        l6 = JernArc(
            start=l1 @ 0, tangent=(l1 % 0).reverse(), radius=hookd / 2, arc_size=170
        )
        l7 = PolarLine(
            start=l6 @ 1,
            length=hookx,
            direction=l6 % 1,
        )
    with BuildSketch(Plane.YZ):
        Circle(radius=maind / 2)
    sweep(path=sprof.wires()[0])
    with BuildLine(mode=Mode.PRIVATE) as stub:
        l7 = Line((0, 0), (0, midx + maind / 2))
    with BuildSketch(Plane.XZ):
        Circle(radius=midd / 2)
    sweep(path=stub.wires()[0])
```

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```
    # splits help keep the object 3d printable by reducing overhang
    split(bisect_by=Plane(origin=(0, 0, -splitz)))
    split(bisect_by=Plane(origin=(0, 0, splitz)), keep=Keep.BOTTOM)

show(mainp)
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show

pegd = 6.35 + 0.1  # mm ~0.25inch
c2c = 25.4  # mm 1.0inch
arcd = 7.2
both = 10
topx = 6
midx = 8
maind = 0.82 * pegd
midd = 1.0 * pegd
hookd = 23
hookx = 10
splitz = maind / 2 - 0.1
topangs = 70

l1 = Line((-both, 0), (c2c - arcd / 2 - 0.5, 0))
l2 = JernArc(start=l1 @ 1, tangent=l1 % 1, radius=arcd / 2, arc_size=topangs)
l3 = PolarLine(
    start=l2 @ 1,
    length=topx,
    direction=l2 % 1,
)
l4 = JernArc(start=l3 @ 1, tangent=l3 % 1, radius=arcd / 2, arc_size=-topangs)
l5 = PolarLine(
    start=l4 @ 1,
    length=topx,
    direction=l4 % 1,
)
l6 = JernArc(start=l1 @ 0, tangent=(l1 % 0).reverse(), radius=hookd / 2, arc_size=170)
l7 = PolarLine(
    start=l6 @ 1,
    length=hookx,
    direction=l6 % 1,
)
sprof = Curve() + (l1, l2, l3, l4, l5, l6, l7)
wire = Wire(sprof.edges())  #  TODO sprof.wires() fails
mainp = sweep(Plane.YZ * Circle(radius=maind / 2), path=wire)

stub = Line((0, 0), (0, midx + maind / 2))
mainp += sweep(Plane.XZ * Circle(radius=midd / 2), path=stub)
```

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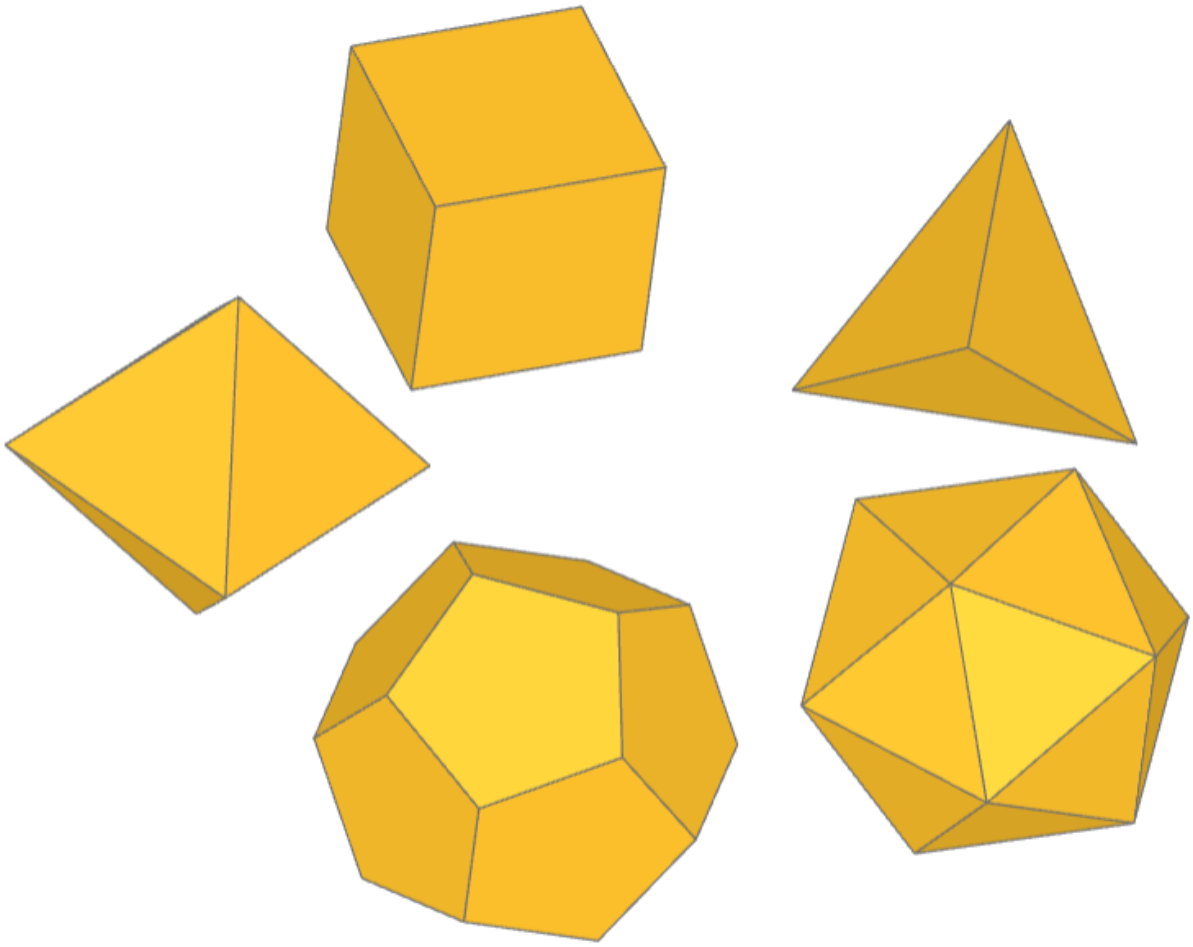
```
# splits help keep the object 3d printable by reducing overhang
mainp = split(mainp, Plane(origin=(0, 0, -splitz)))
mainp = split(mainp, Plane(origin=(0, 0, splitz)), keep=Keep.BOTTOM)

show(mainp)
```

Platonic Solids



This example creates a custom Part object PlatonicSolid.

Platonic solids are five three-dimensional shapes that are highly symmetrical, known since antiquity and named after the ancient Greek philosopher Plato. These solids are unique because their faces are congruent regular polygons, with the same number of faces meeting at each vertex. The five Platonic solids are the tetrahedron (4 triangular faces), cube (6 square faces), octahedron (8 triangular faces), dodecahedron (12 pentagonal faces), and icosahedron (20 triangular faces). Each solid represents a unique way in which identical polygons can be arranged in three dimensions to form a convex polyhedron, embodying ideals of symmetry and balance.

Reference Implementation (Algebra Mode)

```
from build123d import *
from math import sqrt
from typing import Union, Literal
from scipy.spatial import ConvexHull

from ocp_vscode import show

PHI = (1 + sqrt(5)) / 2  # The Golden Ratio


class PlatonicSolid(BasePartObject):
    """Part Object: Platonic Solid

    Create one of the five convex Platonic solids.

    Args:
        face_count (Literal[4,6,8,12,20]): number of faces
        diameter (float): double distance to vertices, i.e. maximum size
        rotation (RotationLike, optional): angles to rotate about axes. Defaults to (0, 0, 0).
        align (Union[None, Align, tuple[Align, Align, Align]], optional): align min, center,
            or max of object. Defaults to None.
        mode (Mode, optional): combine mode. Defaults to Mode.ADD.
    """

    tetrahedron_vertices = [(1, 1, 1), (1, -1, -1), (-1, 1, -1), (-1, -1, 1)]

    cube_vertices = [(i, j, k) for i in [-1, 1] for j in [-1, 1] for k in [-1, 1]]

    octahedron_vertices = (
        [(i, 0, 0) for i in [-1, 1]]
        + [(0, i, 0) for i in [-1, 1]]
        + [(0, 0, i) for i in [-1, 1]]
    )

    dodecahedron_vertices = (
        [(i, j, k) for i in [-1, 1] for j in [-1, 1] for k in [-1, 1]]
        + [(0, i / PHI, j * PHI) for i in [-1, 1] for j in [-1, 1]]
        + [(i / PHI, j * PHI, 0) for i in [-1, 1] for j in [-1, 1]]
        + [(i * PHI, 0, j / PHI) for i in [-1, 1] for j in [-1, 1]]
    )

    icosahedron_vertices = (
        [(0, i, j * PHI) for i in [-1, 1] for j in [-1, 1]]
        + [(i, j * PHI, 0) for i in [-1, 1] for j in [-1, 1]]
        + [(i * PHI, 0, j) for i in [-1, 1] for j in [-1, 1]]
    )

    vertices_lookup = {
        4: tetrahedron_vertices,
        6: cube_vertices,
        8: octahedron_vertices,
```

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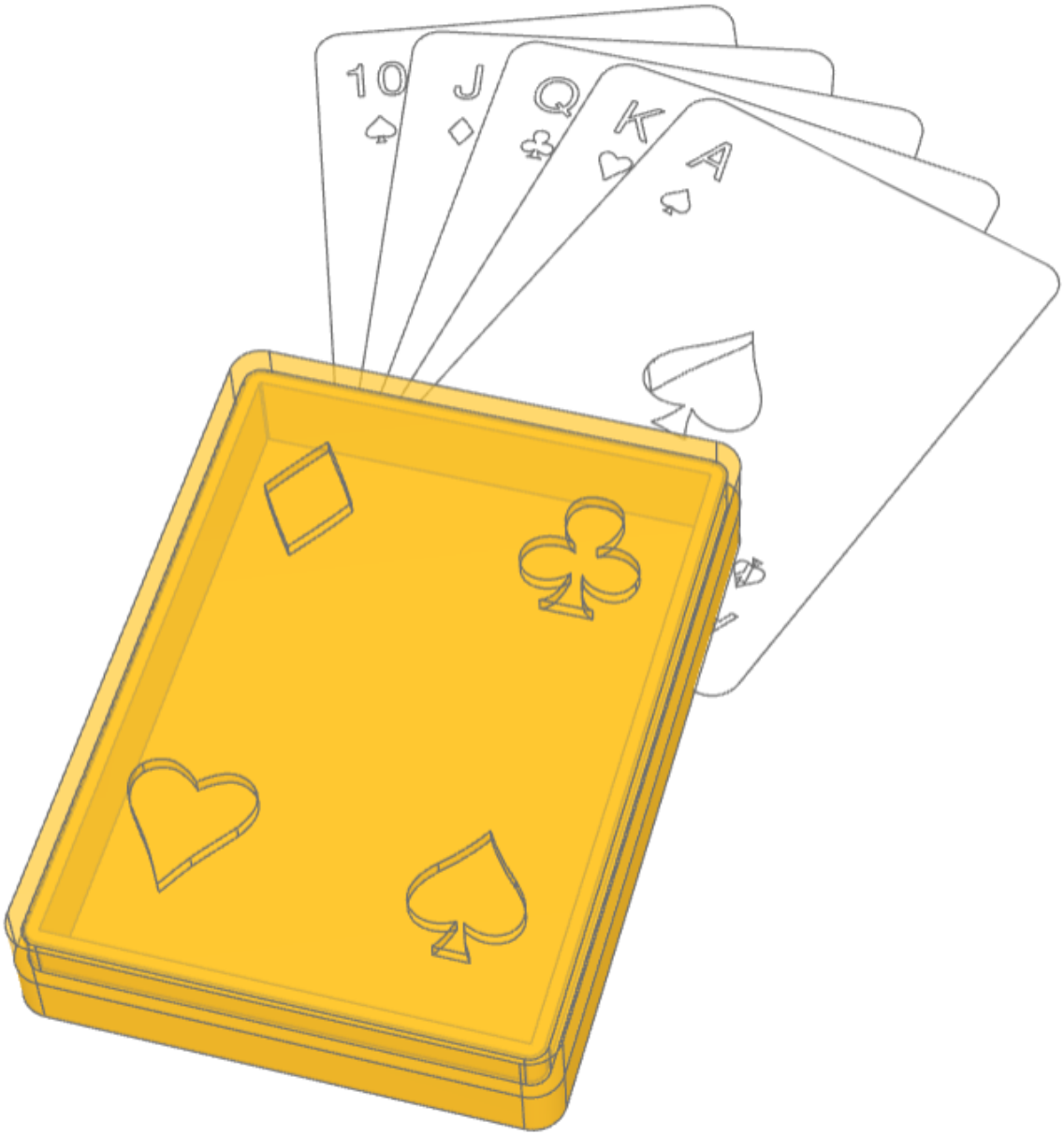
```
        12: dodecahedron_vertices,
        20: icosahedron_vertices,
    }
    _applies_to = [BuildPart._tag]

    def __init__(
        self,
        face_count: Literal[4, 6, 8, 12, 20],
        diameter: float = 1.0,
        rotation: RotationLike = (0, 0, 0),
        align: Union[None, Align, tuple[Align, Align, Align]] = None,
        mode: Mode = Mode.ADD,
    ):
        try:
            platonic_vertices = PlatonicSolid.vertices_lookup[face_count]
        except KeyError:
            raise ValueError(
                f"face_count must be one of 4, 6, 8, 12, or 20 not {face_count}"
            )

        # Create a convex hull from the vertices
        hull = ConvexHull(platonic_vertices).simplices.tolist()

        # Create faces from the vertex indices
        platonic_faces = []
        for face_vertex_indices in hull:
            corner_vertices = [platonic_vertices[i] for i in face_vertex_indices]
            platonic_faces.append(Face(Wire.make_polygon(corner_vertices)))

        # Create the solid from the Faces
        platonic_solid = Solid(Shell(platonic_faces)).clean()

        # By definition, all vertices are the same distance from the origin so
        # scale proportionally to this distance
        platonic_solid = platonic_solid.scale(
            (diameter / 2) / Vector(platonic_solid.vertices()[0]).length
        )

        super().__init__(part=platonic_solid, rotation=rotation, align=align, mode=mode)


solids = [
    Rot(0, 0, 72 * i) * Pos(1, 0, 0) * PlatonicSolid(faces)
    for i, faces in enumerate([4, 6, 8, 12, 20])
]
show(solids)
```

Playing Cards



This example creates a customs Sketch objects: Club, Spade, Heart, Diamond, and PlayingCard in addition to a two part playing card box which has suit cutouts in the lid. The four suits are created with Bézier curves that were imported as code from an SVG file and modified to the code found here.

Reference Implementation (Builder Mode)

```
from typing import Literal
from build123d import *
from ocp_vscode import show_object
```

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```
# [Club]
class Club(BaseSketchObject):
    def __init__(
        self,
        height: float,
        rotation: float = 0,
        align: tuple[Align, Align] = (Align.CENTER, Align.CENTER),
        mode: Mode = Mode.ADD,
    ):
        with BuildSketch() as club:
            with BuildLine():
                l0 = Line((0, -188), (76, -188))
                b0 = Bezier(l0 @ 1, (61, -185), (33, -173), (17, -81))
                b1 = Bezier(b0 @ 1, (49, -128), (146, -145), (167, -67))
                b2 = Bezier(b1 @ 1, (187, 9), (94, 52), (32, 18))
                b3 = Bezier(b2 @ 1, (92, 57), (113, 188), (0, 188))
                mirror(about=Plane.YZ)
            make_face()
            scale(by=height / club.sketch.bounding_box().size.Y)
        super().__init__(obj=club.sketch, rotation=rotation, align=align, mode=mode)


# [Club]


class Spade(BaseSketchObject):
    def __init__(
        self,
        height: float,
        rotation: float = 0,
        align: tuple[Align, Align] = (Align.CENTER, Align.CENTER),
        mode: Mode = Mode.ADD,
    ):
        with BuildSketch() as spade:
            with BuildLine():
                b0 = Bezier((0, 198), (6, 190), (41, 127), (112, 61))
                b1 = Bezier(b0 @ 1, (242, -72), (114, -168), (11, -105))
                b2 = Bezier(b1 @ 1, (31, -174), (42, -179), (53, -198))
                l0 = Line(b2 @ 1, (0, -198))
                mirror(about=Plane.YZ)
            make_face()
            scale(by=height / spade.sketch.bounding_box().size.Y)
        super().__init__(obj=spade.sketch, rotation=rotation, align=align, mode=mode)


class Heart(BaseSketchObject):
    def __init__(
        self,
        height: float,
        rotation: float = 0,
        align: tuple[Align, Align] = (Align.CENTER, Align.CENTER),
```

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```
        mode: Mode = Mode.ADD,
    ):
        with BuildSketch() as heart:
            with BuildLine():
                b1 = Bezier((0, 146), (20, 169), (67, 198), (97, 198))
                b2 = Bezier(b1 @ 1, (125, 198), (151, 186), (168, 167))
                b3 = Bezier(b2 @ 1, (197, 133), (194, 88), (158, 31))
                b4 = Bezier(b3 @ 1, (126, -13), (94, -48), (62, -95))
                b5 = Bezier(b4 @ 1, (40, -128), (0, -198))
                mirror(about=Plane.YZ)
            make_face()
            scale(by=height / heart.sketch.bounding_box().size.Y)
        super().__init__(obj=heart.sketch, rotation=rotation, align=align, mode=mode)


class Diamond(BaseSketchObject):
    def __init__(
        self,
        height: float,
        rotation: float = 0,
        align: tuple[Align, Align] = (Align.CENTER, Align.CENTER),
        mode: Mode = Mode.ADD,
    ):
        with BuildSketch() as diamond:
            with BuildLine():
                Bezier((135, 0), (94, 69), (47, 134), (0, 198))
                mirror(about=Plane.XZ)
                mirror(about=Plane.YZ)
            make_face()
            scale(by=height / diamond.sketch.bounding_box().size.Y)
        super().__init__(obj=diamond.sketch, rotation=rotation, align=align, mode=mode)


card_width = 2.5 * IN
card_length = 3.5 * IN
deck = 0.5 * IN
wall = 4 * MM
gap = 0.5 * MM

with BuildPart() as box_builder:
    with BuildSketch() as plan:
        Rectangle(card_width + 2 * wall, card_length + 2 * wall)
        fillet(plan.vertices(), radius=card_width / 15)
    extrude(amount=wall / 2)
    with BuildSketch(box_builder.faces().sort_by(Axis.Z)[-1]) as walls:
        add(plan.sketch)
        offset(plan.sketch, amount=-wall, mode=Mode.SUBTRACT)
    extrude(amount=deck / 2)
    with BuildSketch(box_builder.faces().sort_by(Axis.Z)[-1]) as inset_walls:
        offset(plan.sketch, amount=-(wall + gap) / 2, mode=Mode.ADD)
        offset(plan.sketch, amount=-wall, mode=Mode.SUBTRACT)
    extrude(amount=deck / 2)
```

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```
with BuildPart() as lid_builder:
    with BuildSketch() as outset_walls:
        add(plan.sketch)
        offset(plan.sketch, amount=-(wall - gap) / 2, mode=Mode.SUBTRACT)
    extrude(amount=deck / 2)
    with BuildSketch(lid_builder.faces().sort_by(Axis.Z)[-1]) as top:
        add(plan.sketch)
    extrude(amount=wall / 2)
    with BuildSketch(lid_builder.faces().sort_by(Axis.Z)[-1]):
        holes = GridLocations(
            3 * card_width / 5, 3 * card_length / 5, 2, 2
        ).local_locations
        for i, hole in enumerate(holes):
            with Locations(hole) as hole_loc:
                if i == 0:
                    Heart(card_length / 5)
                elif i == 1:
                    Diamond(card_length / 5)
                elif i == 2:
                    Spade(card_length / 5)
                elif i == 3:
                    Club(card_length / 5)
    extrude(amount=-wall, mode=Mode.SUBTRACT)

box = Compound(
    [box_builder.part, lid_builder.part.moved(Location((0, 0, (wall + deck) / 2)))]
)
visible, hidden = box.project_to_viewport((70, -50, 120))
max_dimension = max(*Compound(children=visible + hidden).bounding_box().size)
exporter = ExportSVG(scale=100 / max_dimension)
exporter.add_layer("Visible")
exporter.add_layer("Hidden", line_color=(99, 99, 99), line_type=LineType.ISO_DOT)
exporter.add_shape(visible, layer="Visible")
exporter.add_shape(hidden, layer="Hidden")
# exporter.write(f"assets/card_box.svg")


class PlayingCard(BaseSketchObject):
    """PlayingCard

    A standard playing card modelled as a Face.

    Args:
        rank (Literal['A', '2' .. '10', 'J', 'Q', 'K']): card rank
        suit (Literal['Clubs', 'Spades', 'Hearts', 'Diamonds']): card suit
    """

    width = 2.5 * IN
    height = 3.5 * IN
    suits = {"Clubs": Club, "Spades": Spade, "Hearts": Heart, "Diamonds": Diamond}
    ranks = ["A", "2", "3", "4", "5", "6", "7", "8", "9", "10", "J", "Q", "K"]
```

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```
    def __init__(
        self,
        rank: Literal["A", "2", "3", "4", "5", "6", "7", "8", "9", "10", "J", "Q", "K"],
        suit: Literal["Clubs", "Spades", "Hearts", "Diamonds"],
        rotation: float = 0,
        align: tuple[Align, Align] = (Align.CENTER, Align.CENTER),
        mode: Mode = Mode.ADD,
    ):
        with BuildSketch() as playing_card:
            Rectangle(
                PlayingCard.width, PlayingCard.height, align=(Align.MIN, Align.MIN)
            )
            fillet(playing_card.vertices(), radius=PlayingCard.width / 15)
            with Locations(
                (
                    PlayingCard.width / 7,
                    8 * PlayingCard.height / 9,
                )
            ):
                Text(
                    txt=rank,
                    font_size=PlayingCard.width / 7,
                    mode=Mode.SUBTRACT,
                )
            with Locations(
                (
                    PlayingCard.width / 7,
                    7 * PlayingCard.height / 9,
                )
            ):
                PlayingCard.suits[suit](
                    height=PlayingCard.width / 12, mode=Mode.SUBTRACT
                )
            with Locations(
                (
                    6 * PlayingCard.width / 7,
                    1 * PlayingCard.height / 9,
                )
            ):
                Text(
                    txt=rank,
                    font_size=PlayingCard.width / 7,
                    rotation=180,
                    mode=Mode.SUBTRACT,
                )
            with Locations(
                (
                    6 * PlayingCard.width / 7,
                    2 * PlayingCard.height / 9,
                )
            ):
```

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```
                PlayingCard.suits[suit](
                    height=PlayingCard.width / 12, rotation=180, mode=Mode.SUBTRACT
                )
            rank_int = PlayingCard.ranks.index(rank) + 1
            rank_int = rank_int if rank_int < 10 else 1
            with Locations((PlayingCard.width / 2, PlayingCard.height / 2)):
                center_radius = 0 if rank_int == 1 else PlayingCard.width / 3.5
                suit_rotation = 0 if rank_int == 1 else -90
                suit_height = (
                    0.00159 * rank_int**2 - 0.0380 * rank_int + 0.37
                ) * PlayingCard.width
                with PolarLocations(
                    radius=center_radius,
                    count=rank_int,
                    start_angle=90 if rank_int > 1 else 0,
                ):
                    PlayingCard.suits[suit](
                        height=suit_height,
                        rotation=suit_rotation,
                        mode=Mode.SUBTRACT,
                    )
        super().__init__(
            obj=playing_card.sketch, rotation=rotation, align=align, mode=mode
        )


ace_spades = PlayingCard(rank="A", suit="Spades", align=Align.MIN)
ace_spades.color = Color("white")
king_hearts = PlayingCard(rank="K", suit="Hearts", align=Align.MIN)
king_hearts.color = Color("white")
queen_clubs = PlayingCard(rank="Q", suit="Clubs", align=Align.MIN)
queen_clubs.color = Color("white")
jack_diamonds = PlayingCard(rank="J", suit="Diamonds", align=Align.MIN)
jack_diamonds.color = Color("white")
ten_spades = PlayingCard(rank="10", suit="Spades", align=Align.MIN)
ten_spades.color = Color("white")

hand = Compound(
    children=[
        Rot(0, 0, -20) * Pos(0, 0, 0) * ace_spades,
        Rot(0, 0, -10) * Pos(0, 0, -1) * king_hearts,
        Rot(0, 0, 0) * Pos(0, 0, -2) * queen_clubs,
        Rot(0, 0, 10) * Pos(0, 0, -3) * jack_diamonds,
        Rot(0, 0, 20) * Pos(0, 0, -4) * ten_spades,
    ]
)

show_object(Pos(-20, 40) * hand)
show_object(box_builder.part, "box_builder")
show_object(
    Pos(0, 0, (wall + deck) / 2) * lid_builder.part,
    "lid_builder",
```

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```
    options={"alpha": 0.7},
)
```

Stud Wall

This example demonstrates creating custom *Part* objects and putting them into assemblies. The custom object is a *Stud* used in the building industry while the assembly is a *StudWall* created from copies of *Stud* objects for efficiency. Both the *Stud* and *StudWall* objects use *RigidJoints* to define snap points which are used to position all of objects.

Reference Implementation (Algebra Mode)

```
class Stud(BasePartObject):
    """Part Object: Stud

    Create a dimensional framing stud.

    Args:
        length (float): stud size
        width (float): stud size
        thickness (float): stud size
        rotation (RotationLike, optional): angles to rotate about axes. Defaults to (0, 0, 0).
        align (Union[Align, tuple[Align, Align, Align]], optional): align min, center,
            or max of object. Defaults to (Align.CENTER, Align.CENTER, Align.MIN).
        mode (Mode, optional): combine mode. Defaults to Mode.ADD.
    """

    _applies_to = [BuildPart._tag]

    def __init__(
        self,
        length: float = 8 * FT,
        width: float = 3.5 * IN,
        thickness: float = 1.5 * IN,
        rotation: RotationLike = (0, 0, 0),
        align: Union[None, Align, tuple[Align, Align, Align]] = (
            Align.CENTER,
            Align.CENTER,
            Align.MIN,
        ),
        mode: Mode = Mode.ADD,
    ):
        self.length = length
        self.width = width
        self.thickness = thickness

        # Create the basic shape
        with BuildPart() as stud:
            with BuildSketch():
                RectangleRounded(thickness, width, 0.25 * IN)
            extrude(amount=length)

        # Create a Part object with appropriate alignment and rotation
        super().__init__(part=stud.part, rotation=rotation, align=align, mode=mode)

        # Add joints to the ends of the stud
        RigidJoint("end0", self, Location())
        RigidJoint("end1", self, Location((0, 0, length), (1, 0, 0), 180))


class StudWall(Compound):
    """StudWall
```

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```
    A simple stud wall assembly with top and sole plates.

    Args:
        length (float): wall length
        depth (float, optional): stud width. Defaults to 3.5*IN.
        height (float, optional): wall height. Defaults to 8*FT.
        stud_spacing (float, optional): center-to-center. Defaults to 16*IN.
        stud_thickness (float, optional): Defaults to 1.5*IN.
    """

    def __init__(
        self,
        length: float,
        depth: float = 3.5 * IN,
        height: float = 8 * FT,
        stud_spacing: float = 16 * IN,
        stud_thickness: float = 1.5 * IN,
    ):
        # Create the object that will be used for top and sole plates
        plate = Stud(
            length,
            depth,
            rotation=(0, -90, 0),
            align=(Align.MIN, Align.CENTER, Align.MAX),
        )
        # Define where studs will go on the plates
        stud_locations = Pos(stud_thickness / 2, 0, stud_thickness) * GridLocations(
            stud_spacing, 0, int(length / stud_spacing) + 1, 1, align=Align.MIN
        )
        stud_locations.append(Pos(length - stud_thickness / 2, 0, stud_thickness))

        # Create a single stud that will be copied for efficiency
        stud = Stud(height - 2 * stud_thickness, depth, stud_thickness)

        # For efficiency studs in the walls are copies with their own position
        studs = []
        for i, loc in enumerate(stud_locations):
            stud_joint = RigidJoint(f"stud{i}", plate, loc)
            stud_copy = copy.copy(stud)
            stud_joint.connect_to(stud_copy.joints["end0"])
            studs.append(stud_copy)
        top_plate = copy.copy(plate)
        sole_plate = copy.copy(plate)

        # Position the top plate relative to the top of the first stud
        studs[0].joints["end1"].connect_to(top_plate.joints["stud0"])

        # Build the assembly of parts
        super().__init__(children=[top_plate, sole_plate] + studs)

        # Add joints to the wall
        RigidJoint("inside0", self, Location((depth / 2, depth / 2, 0), (0, 0, 1), 90))
```

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```
        RigidJoint("end0", self, Location())


x_wall = StudWall(13 * FT)
y_wall = StudWall(9 * FT)
x_wall.joints["inside0"].connect_to(y_wall.joints["end0"])

show(x_wall, y_wall, render_joints=False)
```

Tea Cup

Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show

wall_thickness = 3 * MM
fillet_radius = wall_thickness * 0.49

with BuildPart() as tea_cup:
    # Create the bowl of the cup as a revolved cross section
    with BuildSketch(Plane.XZ) as bowl_section:
        with BuildLine():
            # Start & end points with control tangents
            s = Spline(
                (30 * MM, 10 * MM),
                (69 * MM, 105 * MM),
                tangents=((1, 0.5), (0.7, 1)),
                tangent_scalars=(1.75, 1),
            )
            # Lines to finish creating ½ the bowl shape
            Polyline(s @ 0, s @ 0 + (10 * MM, -10 * MM), (0, 0), (0, (s @ 1).Y), s @ 1)
        make_face()  # Create a filled 2D shape
    revolve(axis=Axis.Z)
    # Hollow out the bowl with openings on the top and bottom
    offset(amount=-wall_thickness, openings=tea_cup.faces().filter_by(GeomType.PLANE))
    # Add a bottom to the bowl
    with Locations((0, 0, (s @ 0).Y)):
        Cylinder(radius=(s @ 0).X, height=wall_thickness)
    # Smooth out all the edges
    fillet(tea_cup.edges(), radius=fillet_radius)

    # Determine where the handle contacts the bowl
    handle_intersections = [
        tea_cup.part.find_intersection_points(
            Axis(origin=(0, 0, vertical_offset), direction=(1, 0, 0))
        )[-1][0]
        for vertical_offset in [35 * MM, 80 * MM]
    ]
    # Create a path for handle creation
    with BuildLine(Plane.XZ) as handle_path:
        Spline(
            handle_intersections[0] - (wall_thickness / 2, 0),
            handle_intersections[0] + (35 * MM, 30 * MM),
            handle_intersections[0] + (40 * MM, 60 * MM),
            handle_intersections[1] - (wall_thickness / 2, 0),
            tangents=((1, 1.25), (-0.2, -1)),
        )
    # Align the cross section to the beginning of the path
    with BuildSketch(handle_path.line ^ 0) as handle_cross_section:
        RectangleRounded(wall_thickness, 8 * MM, fillet_radius)
    sweep()  # Sweep handle cross section along path

assert abs(tea_cup.part.volume - 130326) < 1
```

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```
show(tea_cup, names=["tea cup"])
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show

wall_thickness = 3 * MM
fillet_radius = wall_thickness * 0.49

# Create the bowl of the cup as a revolved cross section

# Start & end points with control tangents
s = Spline(
    (30 * MM, 10 * MM),
    (69 * MM, 105 * MM),
    tangents=((1, 0.5), (0.7, 1)),
    tangent_scalars=(1.75, 1),
)
# Lines to finish creating ½ the bowl shape
s += Polyline(s @ 0, s @ 0 + (10 * MM, -10 * MM), (0, 0), (0, (s @ 1).Y), s @ 1)
bowl_section = Plane.XZ * make_face(s)  # Create a filled 2D shape
tea_cup = revolve(bowl_section, axis=Axis.Z)

# Hollow out the bowl with openings on the top and bottom
tea_cup = offset(
    tea_cup, -wall_thickness, openings=tea_cup.faces().filter_by(GeomType.PLANE)
)

# Add a bottom to the bowl
tea_cup += Pos(0, 0, (s @ 0).Y) * Cylinder(radius=(s @ 0).X, height=wall_thickness)

# Smooth out all the edges
tea_cup = fillet(tea_cup.edges(), radius=fillet_radius)

# Determine where the handle contacts the bowl
handle_intersections = [
    tea_cup.find_intersection_points(
        Axis(origin=(0, 0, vertical_offset), direction=(1, 0, 0))
    )[-1][0]
    for vertical_offset in [35 * MM, 80 * MM]
]

# Create a path for handle creation
path_spline = Spline(
    handle_intersections[0] - (wall_thickness / 2, 0, 0),
    handle_intersections[0] + (35 * MM, 0, 30 * MM),
    handle_intersections[0] + (40 * MM, 0, 60 * MM),
    handle_intersections[1] - (wall_thickness / 2, 0, 0),
    tangents=((1, 0, 1.25), (-0.2, 0, -1)),
```

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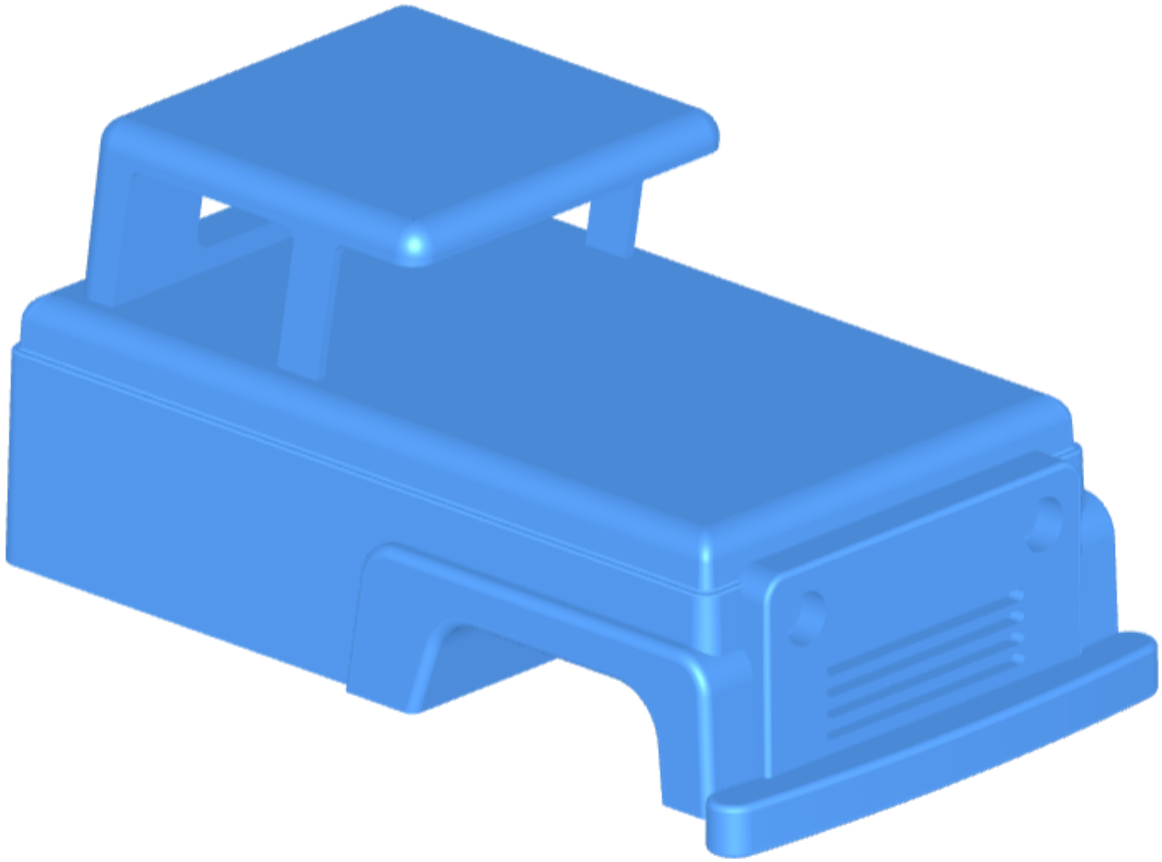
```
)

# Align the cross section to the beginning of the path
location = path_spline ^ 0
handle_cross_section = location * RectangleRounded(wall_thickness, 8 * MM, fillet_radius)

# Sweep handle cross section along path
tea_cup += sweep(handle_cross_section, path=path_spline)

# assert abs(tea_cup.part.volume - 130326.77052487945) < 1e-3

show(tea_cup, names=["tea cup"])
```

This example demonstrates the creation a tea cup, which serves as an example of constructing complex, non-flat geometrical shapes programmatically.

The tea cup model involves several CAD techniques, such as:

- Revolve Operations: There is 1 occurrence of a revolve operation. This is used to create the main body of the tea cup by revolving a profile around an axis, a common technique for generating symmetrical objects like cups.
- Sweep Operations: There are 2 occurrences of sweep operations. The handle are created by sweeping a profile along a path to generate non-planar surfaces.
- Offset/Shell Operations: the bowl of the cup is hollowed out with the offset operation leaving the top open.
- Fillet Operations: There is 1 occurrence of a fillet operation which is used to round the edges for aesthetic improvement and to mimic real-world objects more closely.

Toy Truck



Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show

# Toy Truck Blue
truck_color = Color(0x4683CE)

# Create the main truck body - from bumper to bed, excluding the cab
with BuildPart() as body:
    # The body has two axes of symmetry, so we start with a centered sketch.
    # The default workplane is Plane.XY.
    with BuildSketch() as body_skt:
        Rectangle(20, 35)
        # Fillet all the corners of the sketch.
        # Alternatively, you could use RectangleRounded.
        fillet(body_skt.vertices(), 1)

    # Extrude the body shape upward
    extrude(amount=10, taper=4)
    # Reuse the sketch by accessing it explicitly
    extrude(body_skt.sketch, amount=8, taper=2)

    # Create symmetric fenders on Plane.YZ
    with BuildSketch(Plane.YZ) as fender:
        # The trapezoid has asymmetric angles (80°, 88°)
```

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```
        Trapezoid(18, 6, 80, 88, align=Align.MIN)
        # Fillet top edge vertices (Y-direction highest group)
        fillet(fender.vertices().group_by(Axis.Y)[-1], 1.5)

    # Extrude the fender in both directions
    extrude(amount=10.5, both=True)

    # Create wheel wells with a shifted sketch on Plane.YZ
    with BuildSketch(Plane.YZ.shift_origin((0, 3.5, 0))) as wheel_well:
        Trapezoid(12, 4, 70, 85, align=Align.MIN)
        fillet(wheel_well.vertices().group_by(Axis.Y)[-1], 2)

    # Subtract the wheel well geometry
    extrude(amount=10.5, both=True, mode=Mode.SUBTRACT)

    # Fillet the top edges of the body
    fillet(body.edges().group_by(Axis.Z)[-1], 1)

    # Isolate a set of body edges and preview before filleting
    body_edges = body.edges().group_by(Axis.Z)[-6]
    fillet(body_edges, 0.1)

    # Combine edge groups from both sides of the fender and fillet them
    fender_edges = body.edges().group_by(Axis.X)[0] + body.edges().group_by(Axis.X)[-1]
    fender_edges = fender_edges.group_by(Axis.Z)[1:]
    fillet(fender_edges, 0.4)

    # Create a sketch on the front of the truck for the grill
    with BuildSketch(
        Plane.XZ.offset(-body.vertices().sort_by(Axis.Y)[-1].Y - 0.5)
    ) as grill:
        Rectangle(16, 8.5, align=(Align.CENTER, Align.MIN))
        fillet(grill.vertices().group_by(Axis.Y)[-1], 1)

        # Add headlights (subtractive circles)
        with Locations((0, 6.5)):
            with GridLocations(12, 0, 2, 1):
                Circle(1, mode=Mode.SUBTRACT)

        # Add air vents (subtractive slots)
        with Locations((0, 3)):
            with GridLocations(0, 0.8, 1, 4):
                SlotOverall(10, 0.5, mode=Mode.SUBTRACT)

    # Extrude the grill forward
    extrude(amount=2)

    # Fillet only the outer grill edges (exclude headlight/vent cuts)
    grill_perimeter = body.faces().sort_by(Axis.Y)[-1].outer_wire()
    fillet(grill_perimeter.edges(), 0.2)

    # Create the bumper as a separate part inside the body
```

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```
    with BuildPart() as bumper:
        # Find the midpoint of a front edge and shift slightly to position the bumper
        front_cnt = body.edges().group_by(Axis.Z)[0].sort_by(Axis.Y)[-1] @ 0.5 - (0, 3)

        with BuildSketch() as bumper_plan:
            # Use BuildLine to draw an elliptical arc and offset
            with BuildLine():
                EllipticalCenterArc(front_cnt, 20, 4, start_angle=60, arc_size=60)
                offset(amount=1)
            make_face()

        # Extrude the bumper symmetrically
        extrude(amount=1, both=True)
        fillet(bumper.edges(), 0.25)

    # Define a joint on top of the body to connect the cab later
    RigidJoint("body_top", joint_location=Location((0, -7.5, 10)))
    body.part.color = truck_color

# Create the cab as an independent part to mount on the body
with BuildPart() as cab:
    with BuildSketch() as cab_plan:
        RectangleRounded(16, 16, 1)
        # Split the sketch to work on one symmetric half
        split(bisect_by=Plane.YZ)

    # Extrude the cab forward and upward at an angle
    extrude(amount=7, dir=(0, 0.15, 1))
    fillet(cab.edges().group_by(Axis.Z)[-1].group_by(Axis.X)[1:], 1)

    # Rear window
    with BuildSketch(Plane.XZ.shift_origin((0, 0, 3))) as rear_window:
        RectangleRounded(8, 4, 0.75)
    extrude(amount=10, mode=Mode.SUBTRACT)

    # Front window
    with BuildSketch(Plane.XZ) as front_window:
        RectangleRounded(15.2, 11, 0.75)
    extrude(amount=-10, mode=Mode.SUBTRACT)

    # Side windows
    with BuildSketch(Plane.YZ) as side_window:
        with Locations((3.5, 0)):
            with GridLocations(10, 0, 2, 1):
                Trapezoid(9, 5.5, 80, 100, align=(Align.CENTER, Align.MIN))
                fillet(side_window.vertices().group_by(Axis.Y)[-1], 0.5)
    extrude(amount=12, both=True, mode=Mode.SUBTRACT)

    # Mirror to complete the cab
    mirror(about=Plane.YZ)

    # Define joint on cab base
```

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```
    RigidJoint("cab_base", joint_location=Location((0, 0, 0)))
    cab.part.color = truck_color

# Attach the cab to the truck body using joints
body.joints["body_top"].connect_to(cab.joints["cab_base"])

# Show the result
show(body.part, cab.part)
```

This example demonstrates how to design a toy truck using BuildPart and BuildSketch in Builder mode. The model includes a detailed body, cab, grill, and bumper, showcasing techniques like sketch reuse, symmetry, tapered extrusions, selective filleting, and the use of joints for part assembly. Ideal for learning complex part construction and hierarchical modeling in build123d.

Vase

Reference Implementation (Builder Mode)

```
from build123d import *
from ocp_vscode import show_object

with BuildPart() as vase:
    with BuildSketch() as profile:
        with BuildLine() as outline:
            l1 = Line((0, 0), (12, 0))
            l2 = RadiusArc(l1 @ 1, (15, 20), 50)
            l3 = Spline(l2 @ 1, (22, 40), (20, 50), tangents=(l2 % 1, (-0.75, 1)))
            l4 = RadiusArc(l3 @ 1, l3 @ 1 + Vector(0, 5), 5)
            l5 = Spline(
                l4 @ 1,
                l4 @ 1 + Vector(2.5, 2.5),
                l4 @ 1 + Vector(0, 5),
                tangents=(l4 % 1, (-1, 0)),
            )
            Polyline(
                l5 @ 1,
                l5 @ 1 + Vector(0, 1),
                (0, (l5 @ 1).Y + 1),
                l1 @ 0,
            )
        make_face()
    revolve(axis=Axis.Y)
    offset(openings=vase.faces().filter_by(Axis.Y)[-1], amount=-1)
    top_edges = (
        vase.edges().filter_by_position(Axis.Y, 60, 62).filter_by(GeomType.CIRCLE)
    )
    fillet(top_edges, radius=0.25)
    fillet(vase.edges().sort_by(Axis.Y)[0], radius=0.5)


show_object(Rot(90, 0, 0) * vase.part, name="vase")
```

Reference Implementation (Algebra Mode)

```
from build123d import *
from ocp_vscode import show_object

l1 = Line((0, 0), (12, 0))
l2 = RadiusArc(l1 @ 1, (15, 20), 50)
l3 = Spline(l2 @ 1, (22, 40), (20, 50), tangents=(l2 % 1, (-0.75, 1)))
l4 = RadiusArc(l3 @ 1, l3 @ 1 + Vector(0, 5), 5)
l5 = Spline(
    l4 @ 1,
    l4 @ 1 + Vector(2.5, 2.5),
    l4 @ 1 + Vector(0, 5),
    tangents=(l4 % 1, (-1, 0)),
)
outline = l1 + l2 + l3 + l4 + l5
outline += Polyline(
```

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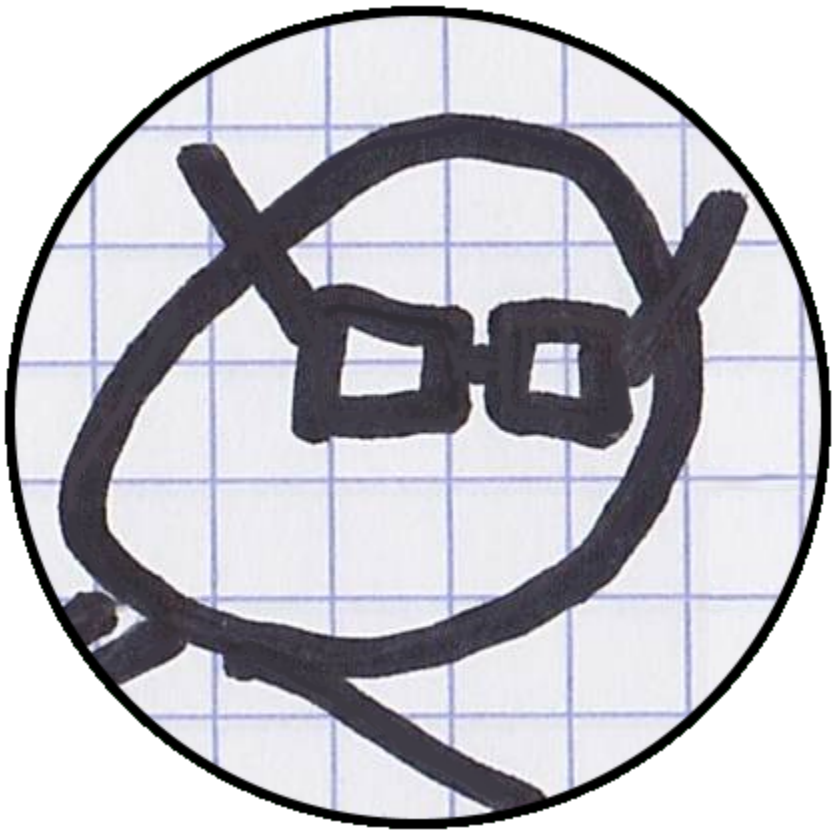
```
    l5 @ 1,
    l5 @ 1 + Vector(0, 1),
    (0, (l5 @ 1).Y + 1),
    l1 @ 0,
)
profile = make_face(outline.edges())
vase = revolve(profile, Axis.Y)
vase = offset(vase, openings=vase.faces().sort_by(Axis.Y).last, amount=-1)

top_edges = vase.edges().filter_by(GeomType.CIRCLE).filter_by_position(Axis.Y, 60, 62)
vase = fillet(top_edges, radius=0.25)

vase = fillet(vase.edges().sort_by(Axis.Y).first, radius=0.5)

show_object(Rot(90, 0, 0) * vase, name="vase")
```

This example demonstrates the build123d techniques involving the creation of a vase. Specifically, it showcases the processes of revolving a sketch, shelling (creating a hollow object by removing material from its interior), and selecting edges by position range and type for the application of fillets (rounding off the edges).

- Sketching: Drawing a 2D profile or outline that represents the side view of the vase.
- Revolving: Rotating the sketch around an axis to create a 3D object. This step transforms the 2D profile into a 3D vase shape.
- Offset/Shelling: Removing material from the interior of the solid vase to create a hollow space, making it resemble a real vase more closely.
- Edge Filleting: Selecting specific edges of the vase for filleting, which involves rounding those edges. The edges are selected based on their position and type.

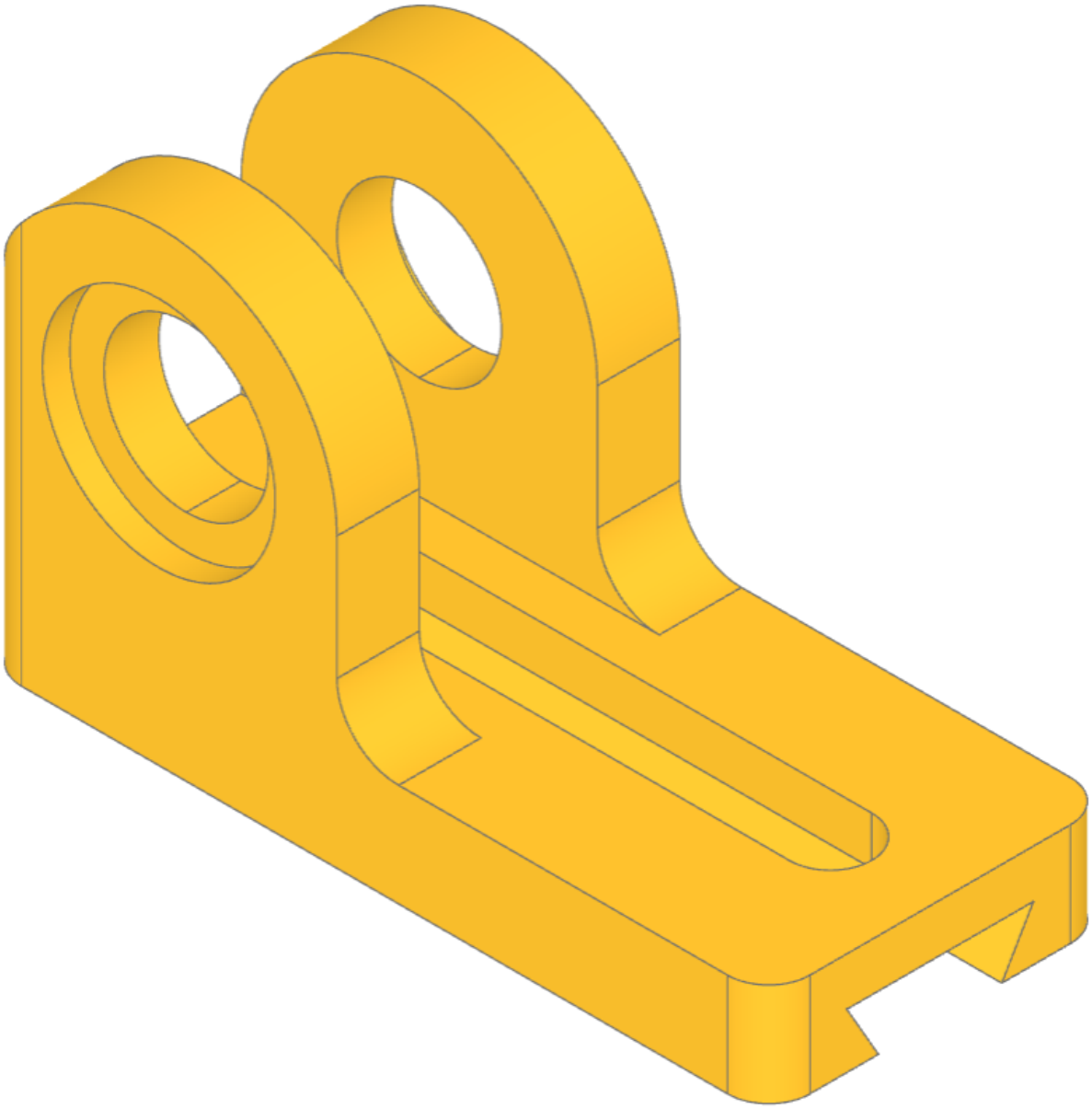
1.9.7 Too Tall Toby (TTT) Tutorials



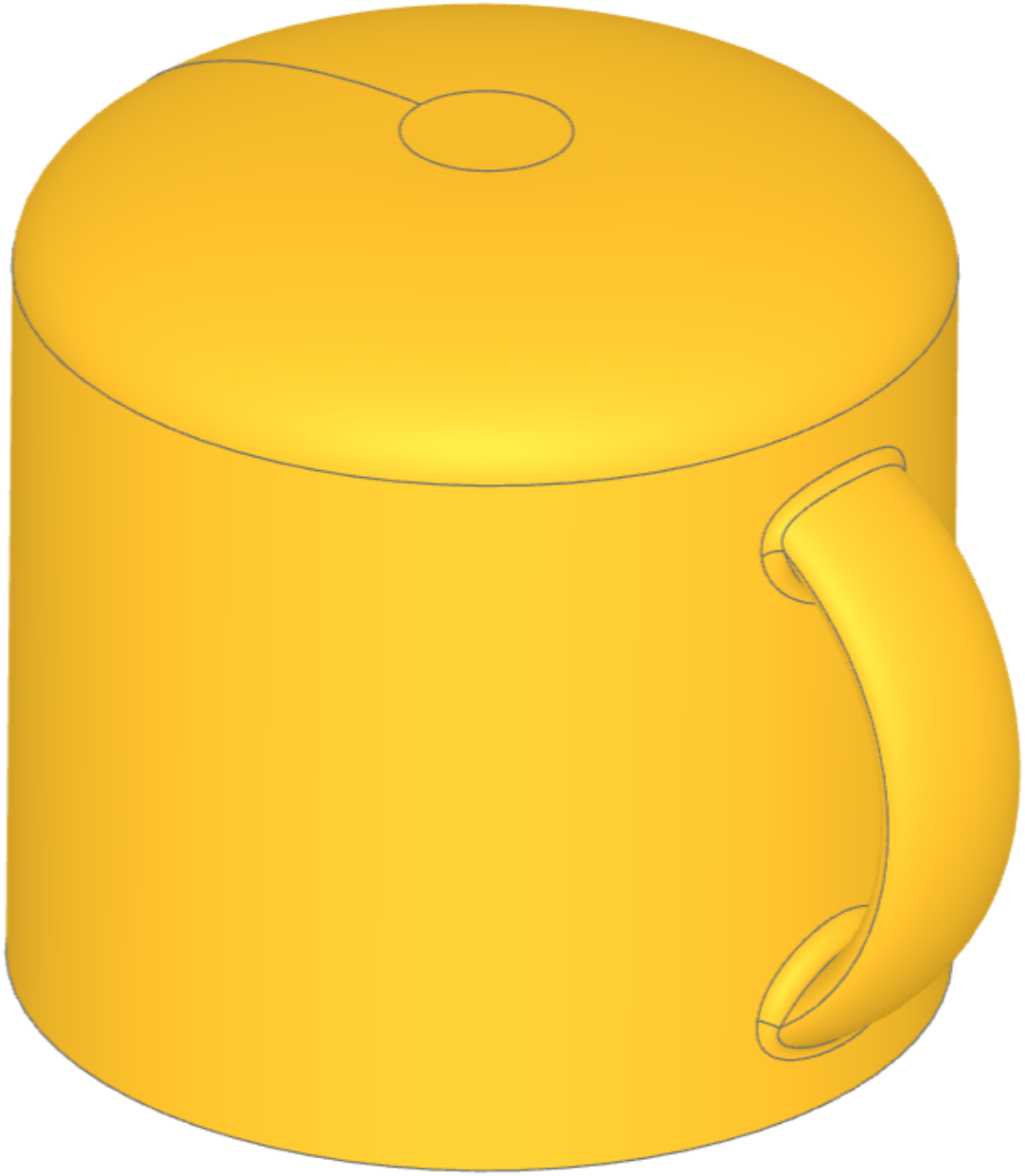
To enhance users' proficiency with Build123D, this section offers a series of challenges. In these challenges, users are presented with a CAD drawing and tasked with designing the part. Their goal is to match the part's mass to a specified target.

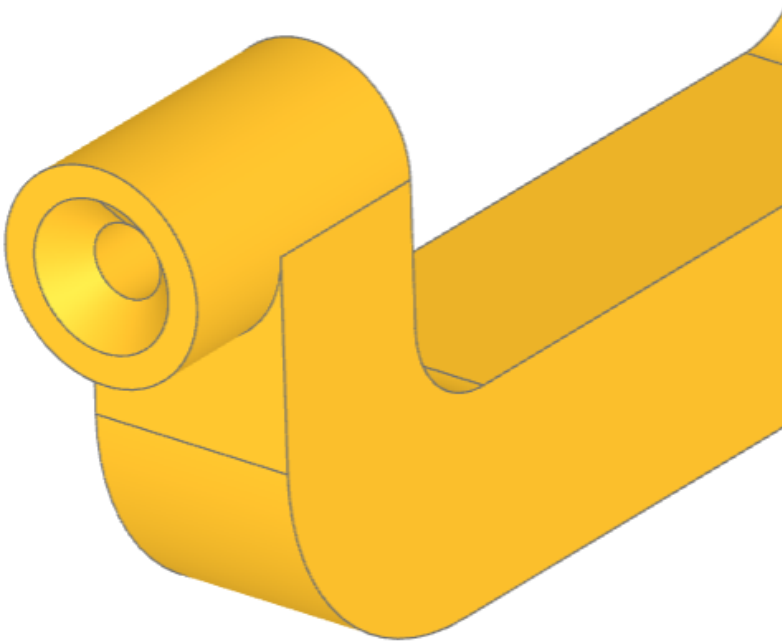
These drawings were skillfully crafted and generously provided to Build123D by Too Tall Toby, a renowned figure in the realm of 3D CAD. Too Tall Toby is the host of the World Championship of 3D CAD Speedmodeling. For additional 3D CAD challenges and content, be sure to visit Toby's youtube channel.

Feel free to click on the parts below to embark on these engaging challenges.

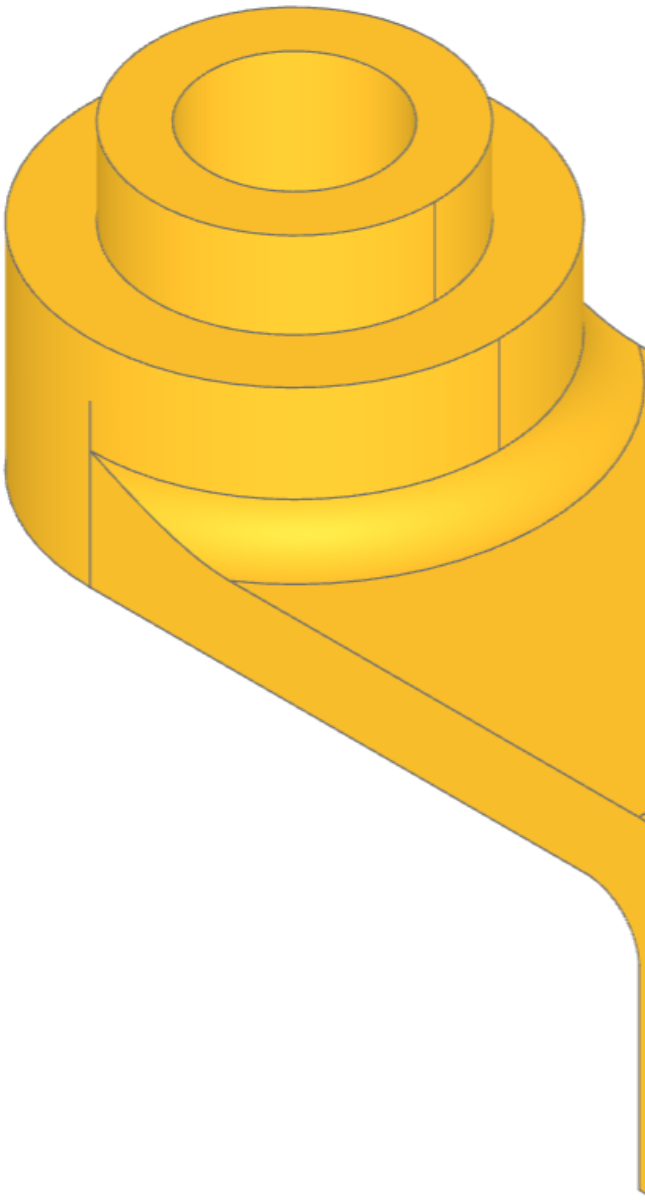


Party Pack 01-01 Bearing Bracket *Party Pack 01-01 Bearing Bracket*

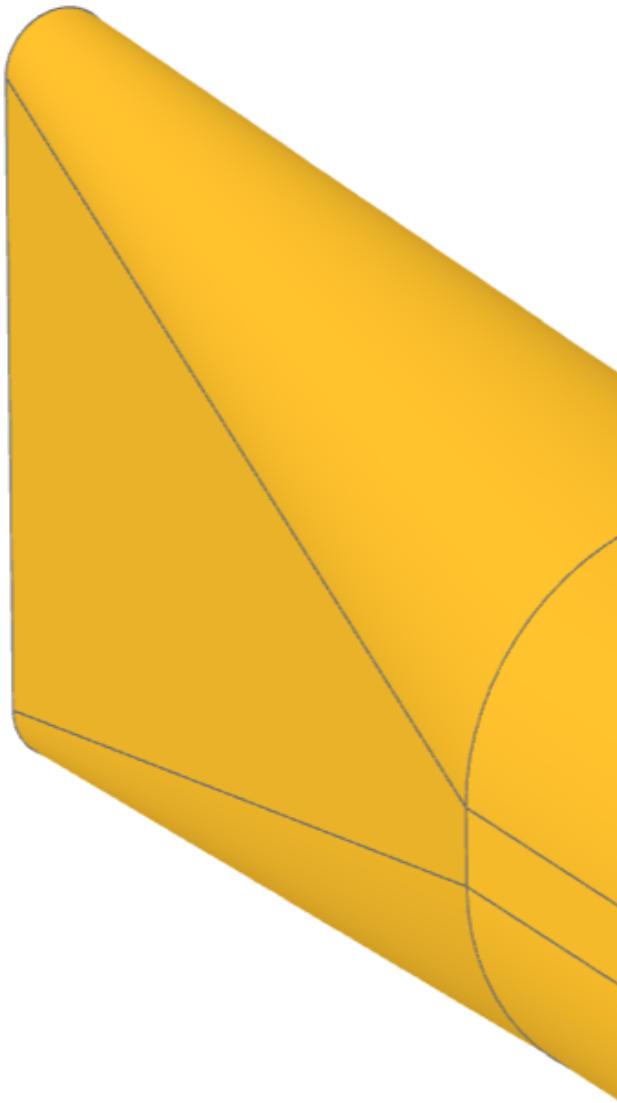




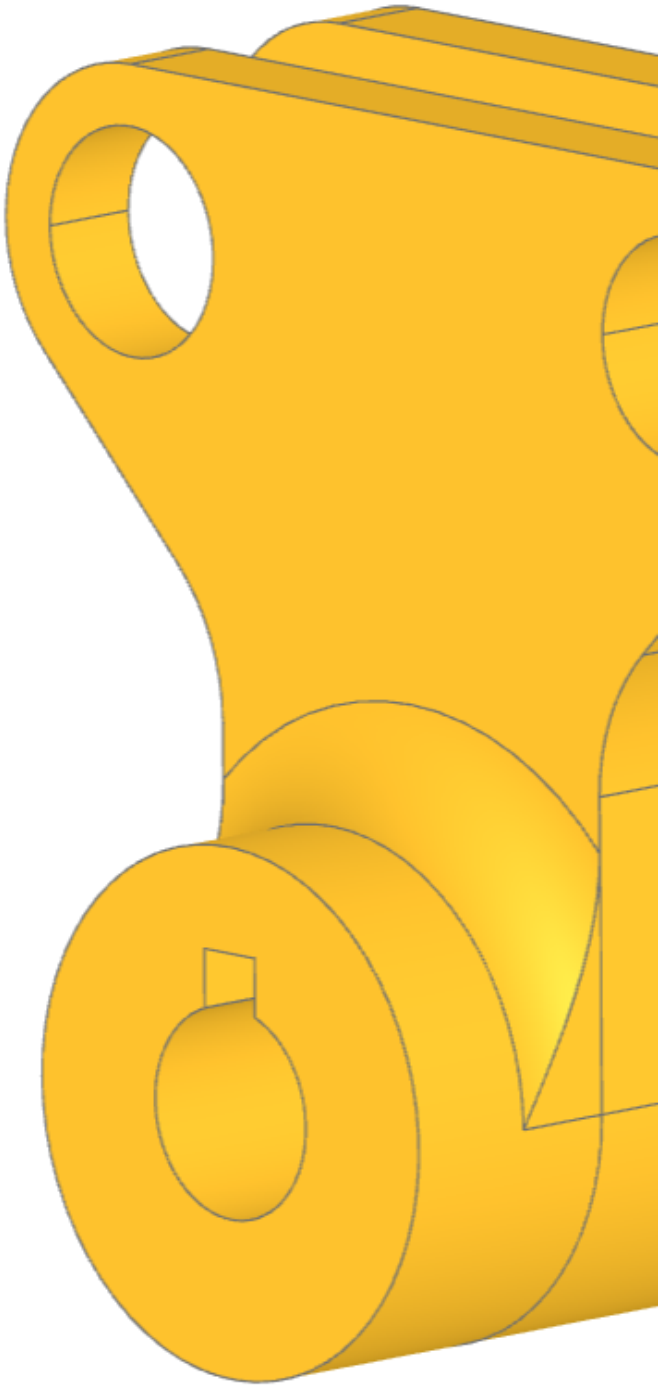
Party Pack 01-02 Post Cap *Party Pack 01-02 Post Cap*



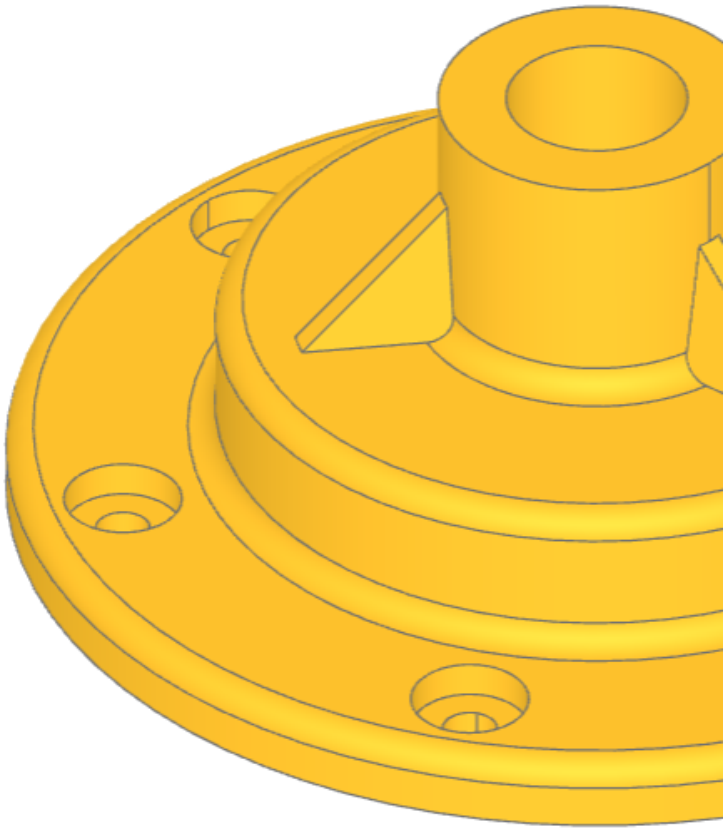
Party Pack 01-03 C Clamp Base *Party Pack 01-03 C Clamp Base*



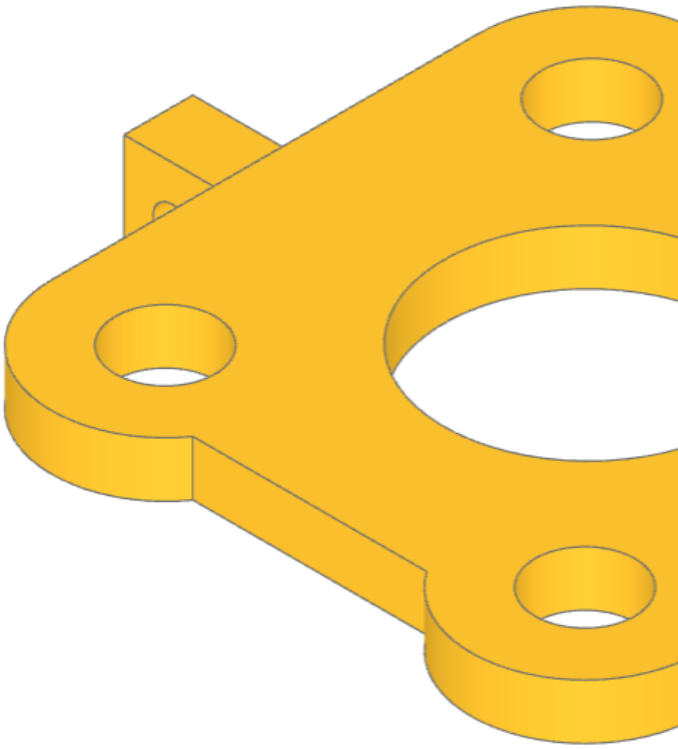
Party Pack 01-04 Angle Bracket *Party Pack 01-04 Angle Bracket*



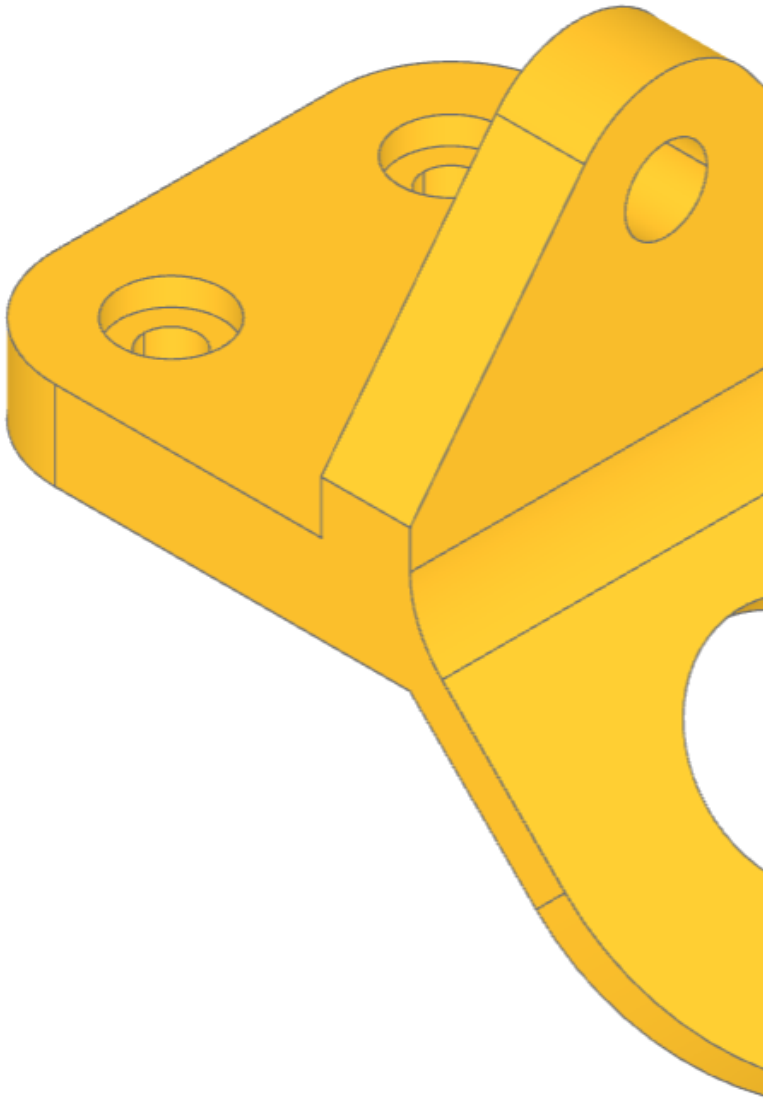
Party Pack 01-05 Paste Sleeve *Party Pack 01-05 Paste Sleeve*



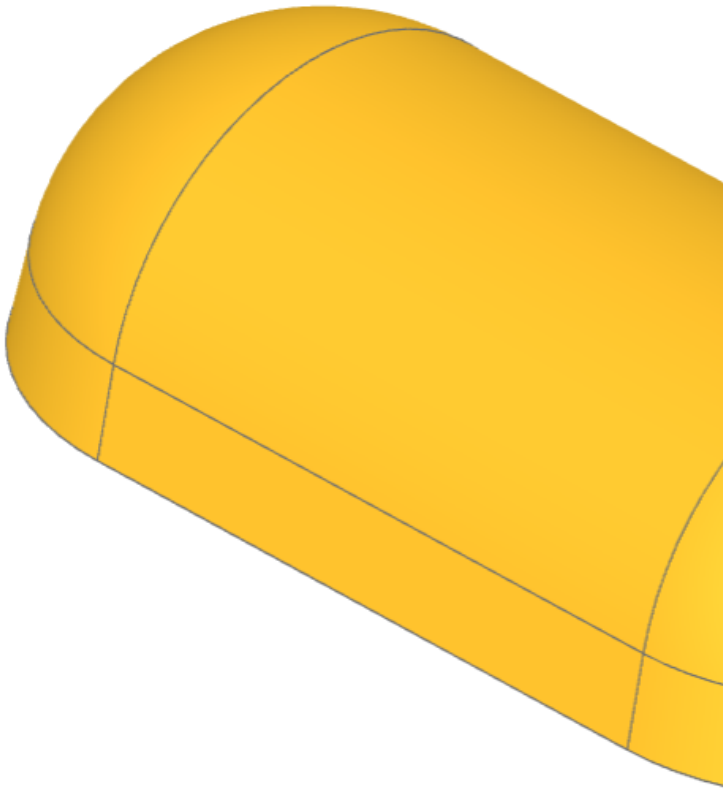
Party Pack 01-06 Bearing Jig *Party Pack 01-06 Bearing Jig*



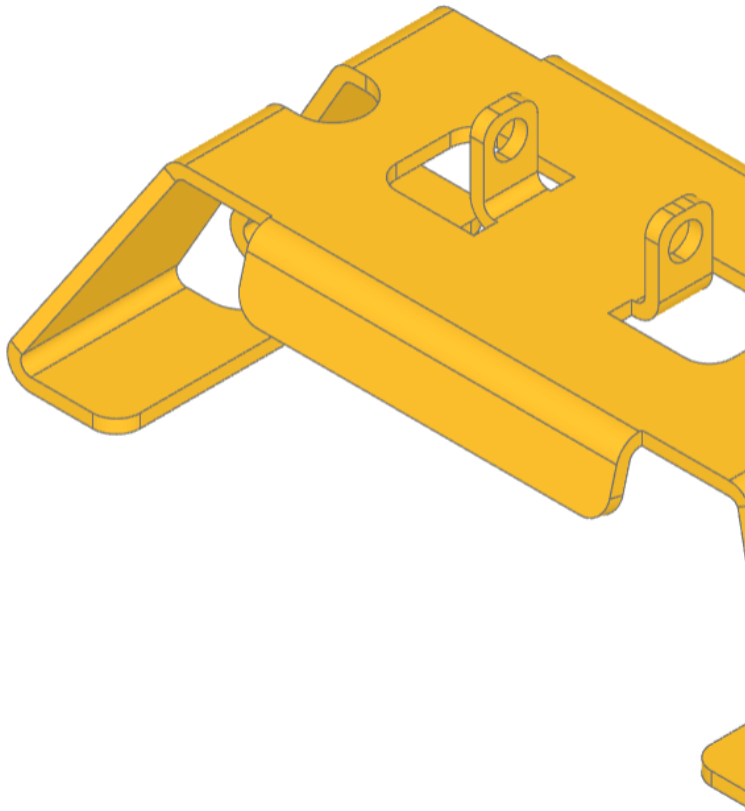
Party Pack 01-07 Flanged Hub *Party Pack 01-07 Flanged Hub*



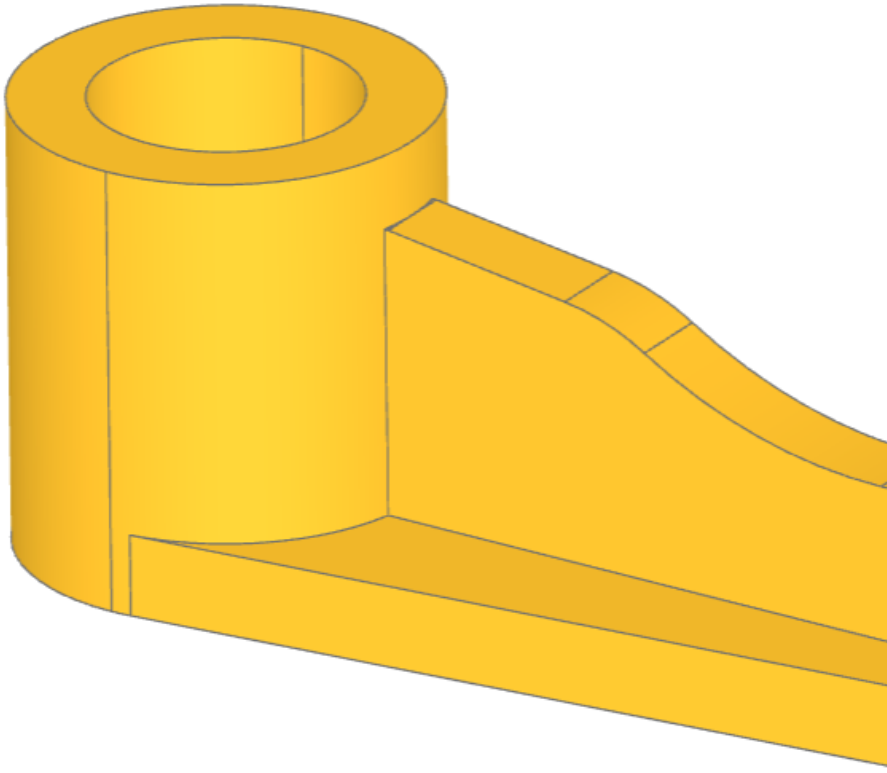
Party Pack 01-08 Tie Plate *Party Pack 01-08 Tie Plate*



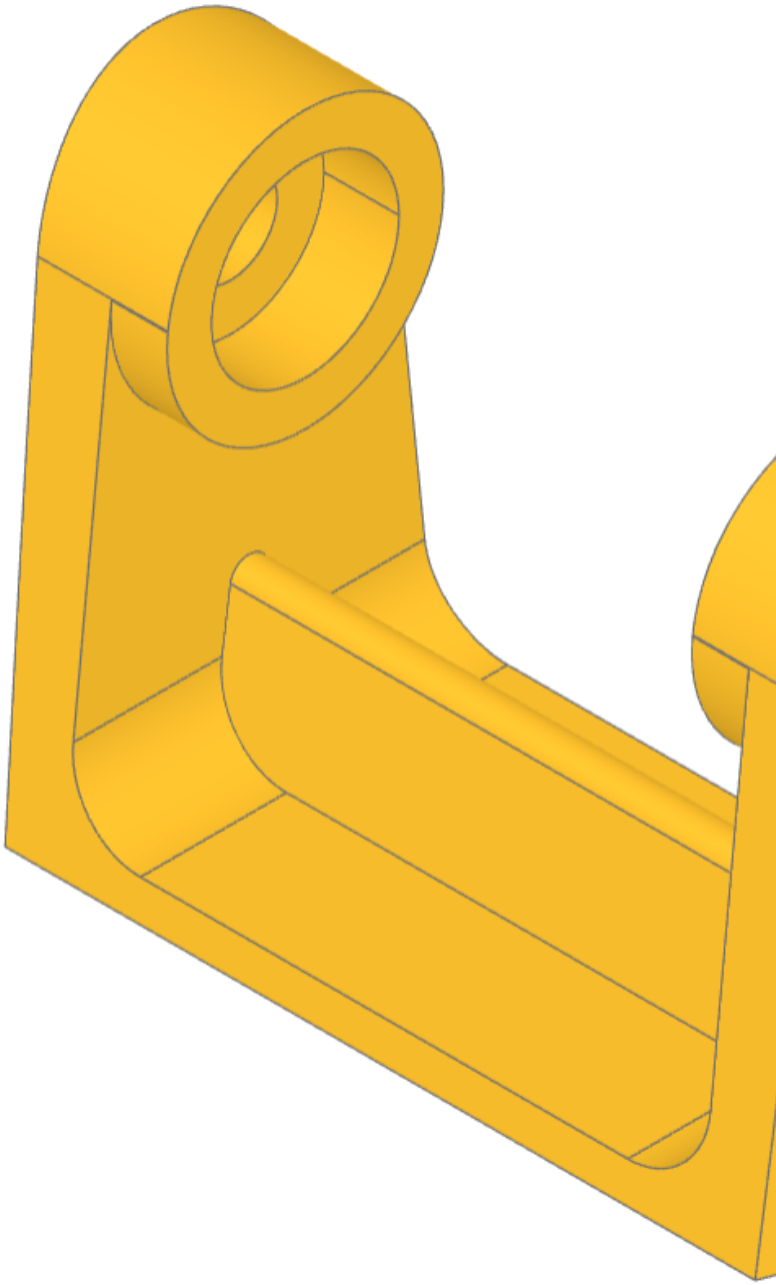
Party Pack 01-09 Corner Tie *Party Pack 01-09 Corner Tie*



Party Pack 01-10 Light Cap *Party Pack 01-10 Light Cap*



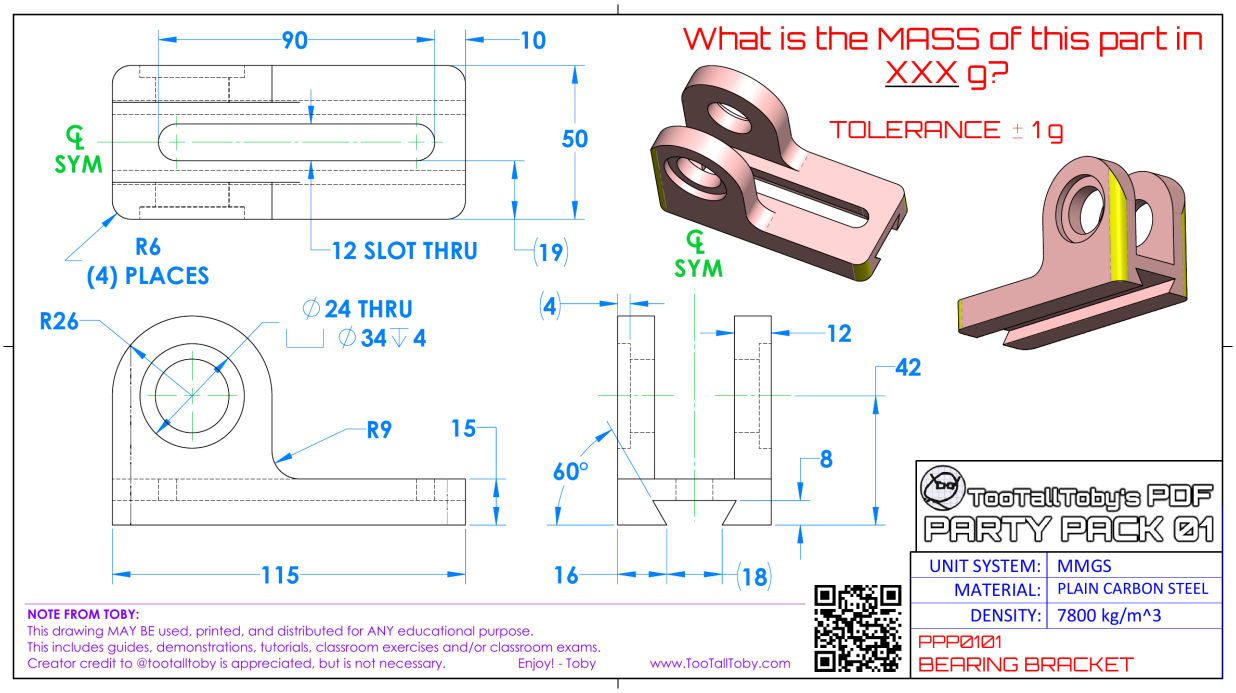
23-02-02 SM Hanger *23-02-02 SM Hanger*



23-T-24 Curved Support *23-T-24 Curved Support*

24-SPO-06 Buffer Stand *24-SPO-06 Buffer Stand*

Party Pack 01-01 Bearing Bracket



Object Mass

797.15 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-01 Bearing Bracket
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS

with BuildPart() as p:
    with BuildSketch() as s:
        Rectangle(115, 50)
        with Locations((5 / 2, 0)):
            SlotOverall(90, 12, mode=Mode.SUBTRACT)
    extrude(amount=15)

    with BuildSketch(Plane.XZ.offset(50 / 2)) as s3:
        with Locations((-115 / 2 + 26, 15)):
            SlotOverall(42 + 2 * 26 + 12, 2 * 26, rotation=90)
    zz = extrude(amount=-12)
    split(bisect_by=Plane.XY)
```

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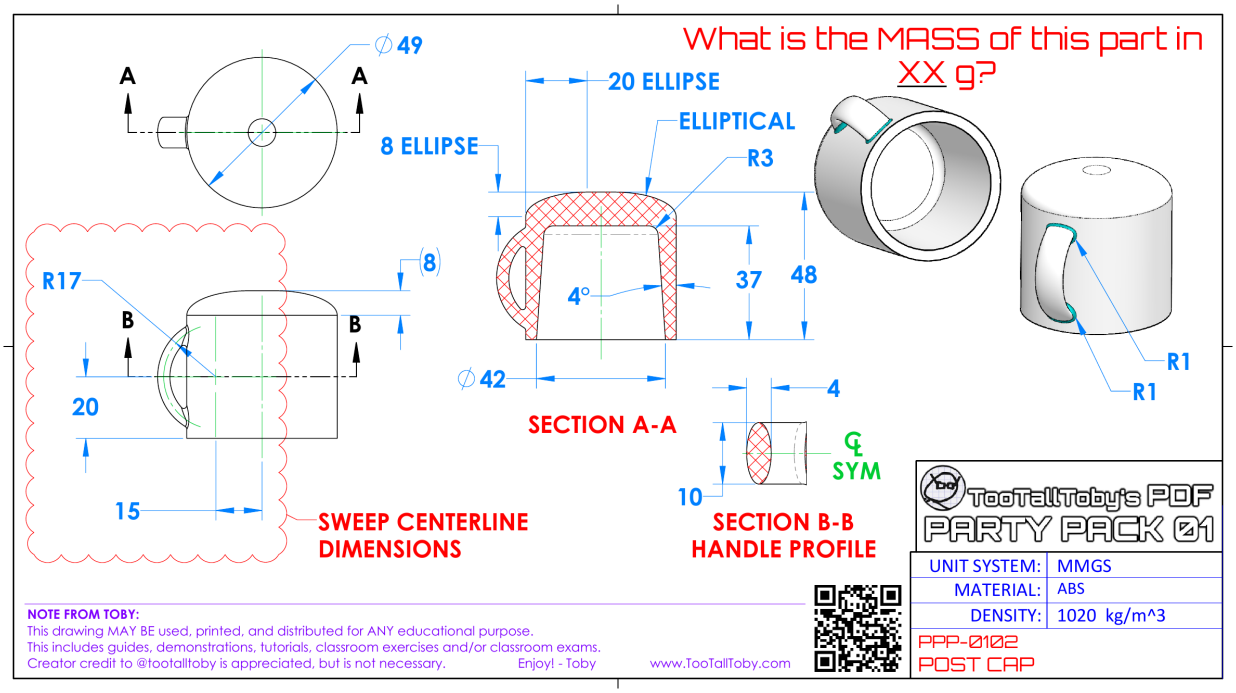
```
    edgs = p.part.edges().filter_by(Axis.Y).group_by(Axis.X)[-2]
    fillet(edgs, 9)

    with Locations(zz.faces().sort_by(Axis.Y)[0]):
        with Locations((42 / 2 + 6, 0)):
            CounterBoreHole(24 / 2, 34 / 2, 4)
    mirror(about=Plane.XZ)

    with BuildSketch() as s4:
        RectangleRounded(115, 50, 6)
    extrude(amount=80, mode=Mode.INTERSECT)
    # fillet does not work right, mode intersect is safer

    with BuildSketch(Plane.YZ) as s4:
        with BuildLine() as bl:
            l1 = Line((0, 0), (18 / 2, 0))
            l2 = PolarLine(l1 @ 1, 8, 60, length_mode=LengthMode.VERTICAL)
            l3 = Line(l2 @ 1, (0, 8))
            mirror(about=Plane.YZ)
        make_face()
    extrude(amount=115/2, both=True, mode=Mode.SUBTRACT)

show_object(p)


got_mass = p.part.volume*densa
want_mass = 797.15
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-02 Post Cap



Object Mass

43.09 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-02 Post Cap
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS


# TTT Party Pack 01: PPP0102, mass(abs) = 43.09g
with BuildPart() as p:
    with BuildSketch(Plane.XZ) as sk1:
        Rectangle(49, 48 - 8, align=(Align.CENTER, Align.MIN))
        Rectangle(9, 48, align=(Align.CENTER, Align.MIN))
        with Locations((9 / 2, 40)):
            Ellipse(20, 8)
        split(bisect_by=Plane.YZ)
    revolve(axis=Axis.Z)

    with BuildSketch(Plane.YZ.offset(-15)) as xc1:
```

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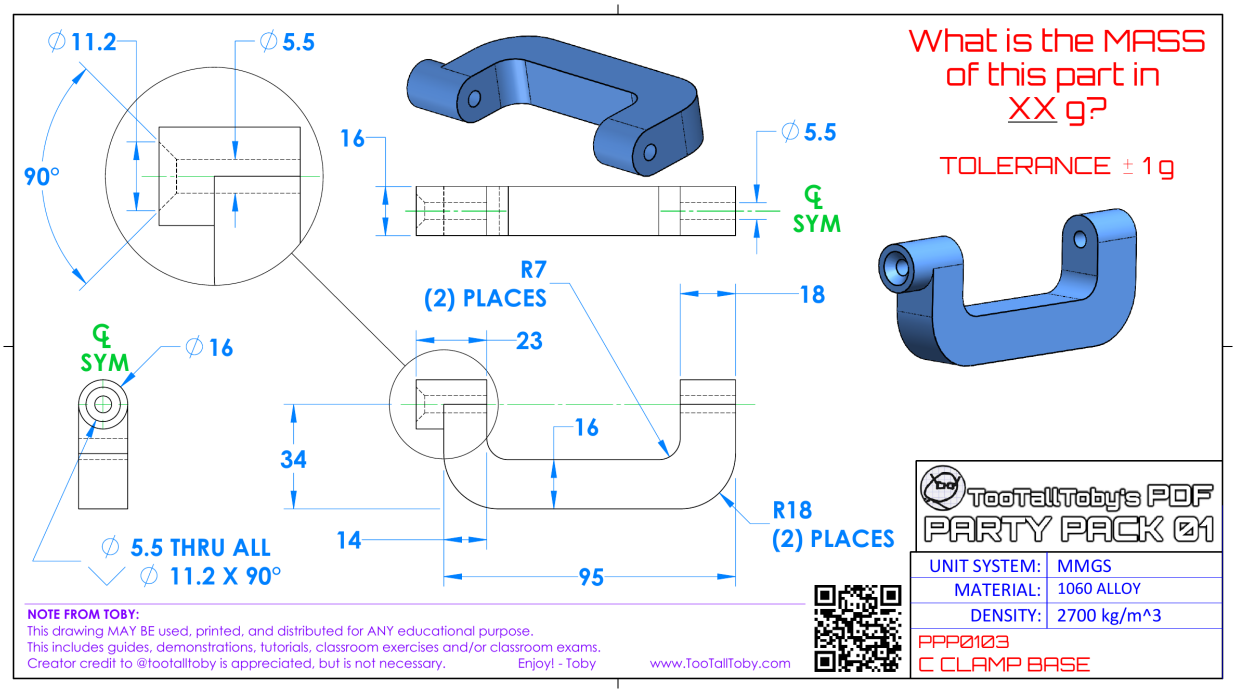
```
        with Locations((0, 40 / 2 - 17)):
            Ellipse(10 / 2, 4 / 2)
        with BuildLine(Plane.XZ) as l1:
            CenterArc((-15, 40 / 2), 17, 90, 180)
    sweep(path=l1)

    fillet(p.edges().filter_by(GeomType.CIRCLE, reverse=True).group_by(Axis.X)[0], 1)

    with BuildLine(mode=Mode.PRIVATE) as lc1:
        PolarLine(
            (42 / 2, 0), 37, 94, length_mode=LengthMode.VERTICAL
        )  # construction line

    pts = [
        (0, 0),
        (42 / 2, 0),
        ((lc1.line @ 1).X, (lc1.line @ 1).Y),
        (0, (lc1.line @ 1).Y),
    ]
    with BuildSketch(Plane.XZ) as sk2:
        Polygon(*pts, align=None)
        fillet(sk2.vertices().group_by(Axis.X)[1], 3)
    revolve(axis=Axis.Z, mode=Mode.SUBTRACT)

show(p)


got_mass = p.part.volume*densc
want_mass = 43.09
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-03 C Clamp Base



Object Mass

96.13 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-03 C Clamp Base
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS


with BuildPart() as ppp0103:
    with BuildSketch() as sk1:
        RectangleRounded(34 * 2, 95, 18)
        with Locations((0, -2)):
            RectangleRounded((34 - 16) * 2, 95 - 18 - 14, 7, mode=Mode.SUBTRACT)
        with Locations((-34 / 2, 0)):
            Rectangle(34, 95, 0, mode=Mode.SUBTRACT)
    extrude(amount=16)
    with BuildSketch(Plane.XZ.offset(-95 / 2)) as cyl1:
        with Locations((0, 16 / 2)):
            Circle(16 / 2)
```

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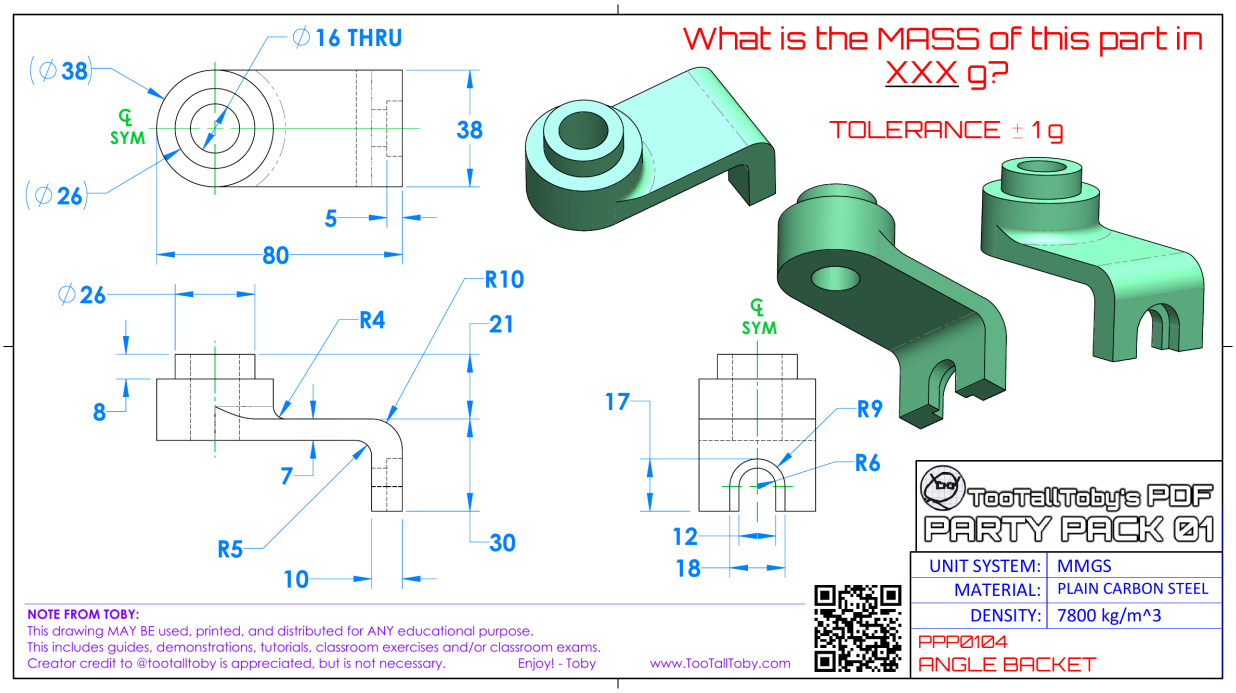
```
    extrude(amount=18)
    with BuildSketch(Plane.XZ.offset(95 / 2 - 14)) as cyl2:
        with Locations((0, 16 / 2)):
            Circle(16 / 2)
    extrude(amount=23)
    with Locations(Plane.XZ.offset(95 / 2 + 9)):
        with Locations((0, 16 / 2)):
            CounterSinkHole(5.5 / 2, 11.2 / 2, None, 90)

show(ppp0103)

got_mass = ppp0103.part.volume*densb
want_mass = 96.13
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-04 Angle Bracket



Object Mass

310.00 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-04 Angle Bracket
"""
```

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```
from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS

d1, d2, d3 = 38, 26, 16
h1, h2, h3, h4 = 20, 8, 7, 23
w1, w2, w3 = 80, 10, 5
f1, f2, f3 = 4, 10, 5
sloth1, sloth2 = 18, 12
slotw1, slotw2 = 17, 14

with BuildPart() as p:
    with BuildSketch() as s:
        Circle(d1 / 2)
    extrude(amount=h1)
    with BuildSketch(Plane.XY.offset(h1)) as s2:
        Circle(d2 / 2)
    extrude(amount=h2)
    with BuildSketch(Plane.YZ) as s3:
        Rectangle(d1 + 15, h3, align=(Align.CENTER, Align.MIN))
    extrude(amount=w1 - d1 / 2)
    # fillet workaround \/
    ped = p.part.edges().group_by(Axis.Z)[2].filter_by(GeomType.CIRCLE)
    fillet(ped, f1)
    with BuildSketch(Plane.YZ) as s3a:
        Rectangle(d1 + 15, 15, align=(Align.CENTER, Align.MIN))
        Rectangle(d1, 15, mode=Mode.SUBTRACT, align=(Align.CENTER, Align.MIN))
    extrude(amount=w1 - d1 / 2, mode=Mode.SUBTRACT)
    # end fillet workaround /\
    with BuildSketch() as s4:
        Circle(d3 / 2)
    extrude(amount=h1 + h2, mode=Mode.SUBTRACT)
    with BuildSketch() as s5:
        with Locations((w1 - d1 / 2 - w2 / 2, 0)):
            Rectangle(w2, d1)
    extrude(amount=-h4)
    fillet(p.part.edges().group_by(Axis.X)[-1].sort_by(Axis.Z)[-1], f2)
    fillet(p.part.edges().group_by(Axis.X)[-4].sort_by(Axis.Z)[-2], f3)
    pln = Plane.YZ.offset(w1 - d1 / 2)
    with BuildSketch(pln) as s6:
        with Locations((0, -h4)):
            SlotOverall(slotw1 * 2, sloth1, 90)
    extrude(amount=-w3, mode=Mode.SUBTRACT)
    with BuildSketch(pln) as s6b:
        with Locations((0, -h4)):
            SlotOverall(slotw2 * 2, sloth2, 90)
    extrude(amount=-w2, mode=Mode.SUBTRACT)
```

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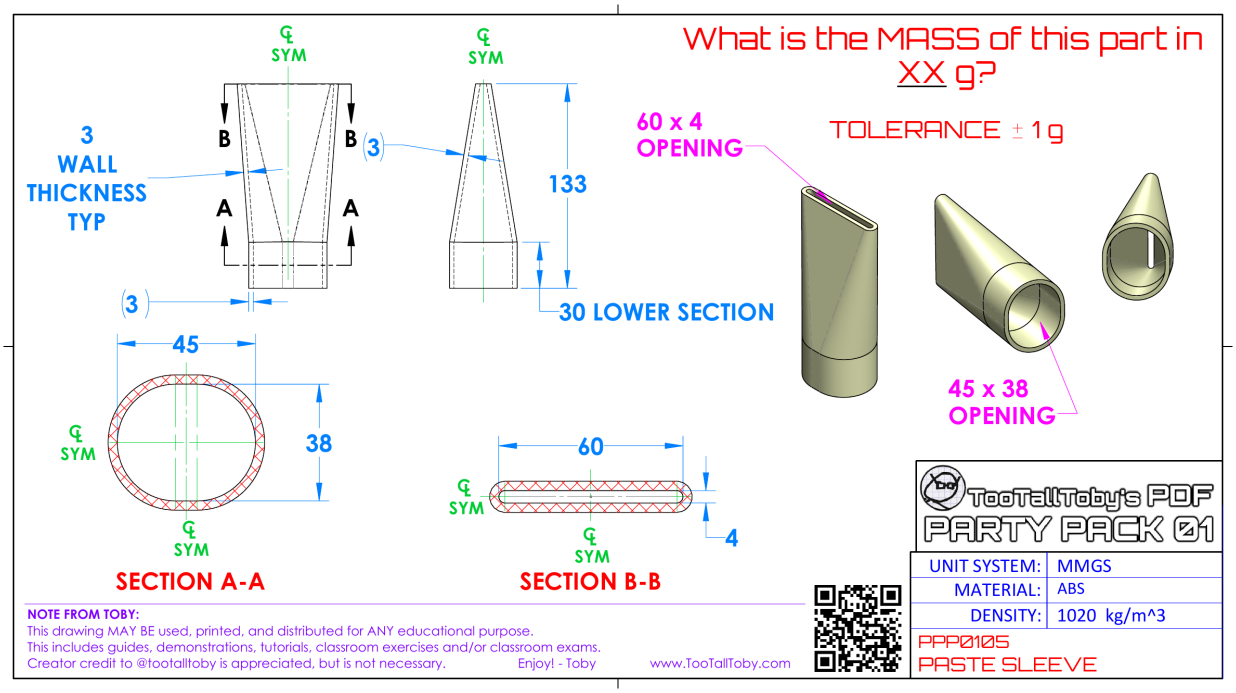
```
show(p)


got_mass = p.part.volume*densa
want_mass = 310
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-05 Paste Sleeve



Object Mass

57.08 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-05 Paste Sleeve
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS
```

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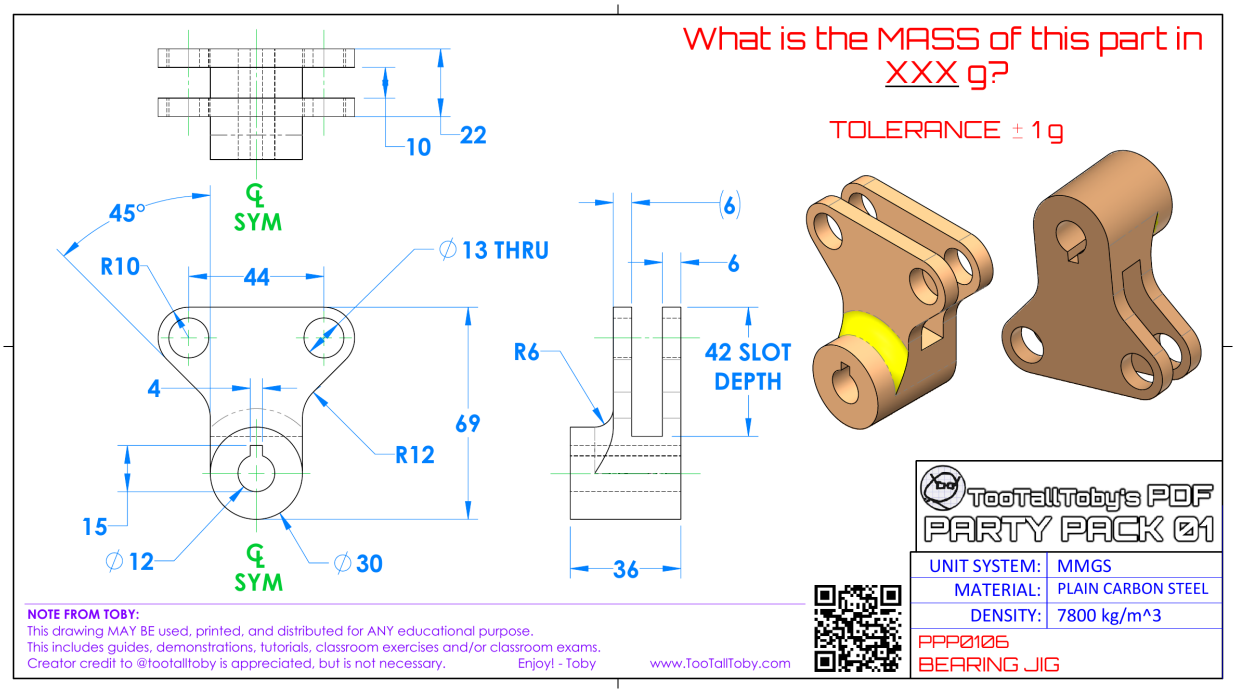
```
with BuildPart() as p:
    with BuildSketch() as s:
        SlotOverall(45, 38)
        offset(amount=3)
    with BuildSketch(Plane.XY.offset(133 - 30)) as s2:
        SlotOverall(60, 4)
        offset(amount=3)
    loft()

    with BuildSketch() as s3:
        SlotOverall(45, 38)
    with BuildSketch(Plane.XY.offset(133 - 30)) as s4:
        SlotOverall(60, 4)
    loft(mode=Mode.SUBTRACT)

    extrude(p.part.faces().sort_by(Axis.Z)[0], amount=30)

show(p)


got_mass = p.part.volume*densc
want_mass = 57.08
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-06 Bearing Jig



Object Mass

328.02 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-06 Bearing Jig
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS

r1, r2, r3, r4, r5 = 30 / 2, 13 / 2, 12 / 2, 10, 6  # radii used
x1 = 44  # lengths used
y1, y2, y3, y4, y_tot = 36, 36 - 22 / 2, 22 / 2, 42, 69  # widths used

with BuildSketch(Location((0, -r1, y3))) as sk_body:
    with BuildLine() as l:
        c1 = Line((r1, 0), (r1, y_tot), mode=Mode.PRIVATE)  # construction line
        m1 = Line((0, y_tot), (x1 / 2, y_tot))
        m2 = JernArc(m1 @ 1, m1 % 1, r4, -90 - 45)
        m3 = IntersectingLine(m2 @ 1, m2 % 1, c1)
        m4 = Line(m3 @ 1, (r1, r1))
        m5 = JernArc(m4 @ 1, m4 % 1, r1, -90)
        mirror(about=Plane.YZ)
    make_face()
    fillet(sk_body.vertices().group_by(Axis.Y)[1], 12)
    with Locations((x1 / 2, y_tot - 10), (-x1 / 2, y_tot - 10)):
        Circle(r2, mode=Mode.SUBTRACT)
    # Keyway
    with Locations((0, r1)):
        Circle(r3, mode=Mode.SUBTRACT)
        Rectangle(4, 3 + 6, align=(Align.CENTER, Align.MIN), mode=Mode.SUBTRACT)

with BuildPart() as p:
    Box(200, 200, 22)  # Oversized plate
    # Cylinder underneath
    Cylinder(r1, y2, align=(Align.CENTER, Align.CENTER, Align.MAX))
    fillet(p.edges(Select.NEW), r5)  # Weld together
    extrude(sk_body.sketch, amount=-y1, mode=Mode.INTERSECT)  # Cut to shape
    # Remove slot
    with Locations((0, y_tot - r1 - y4, 0)):
        Box(
            y_tot,
            y_tot,
            10,
            align=(Align.CENTER, Align.MIN, Align.CENTER),
            mode=Mode.SUBTRACT,
        )
```

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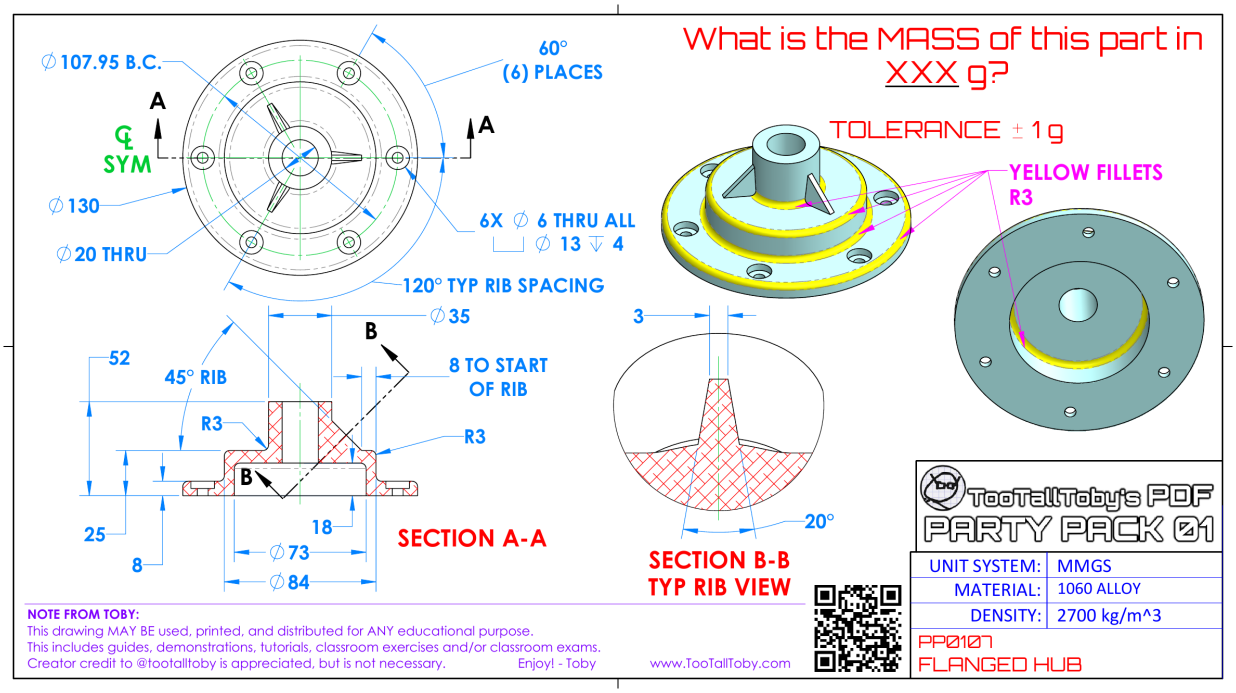
```
show(p)


got_mass = p.part.volume*densa
want_mass = 328.02
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-07 Flanged Hub



Object Mass

372.99 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-07 Flanged Hub
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS
```

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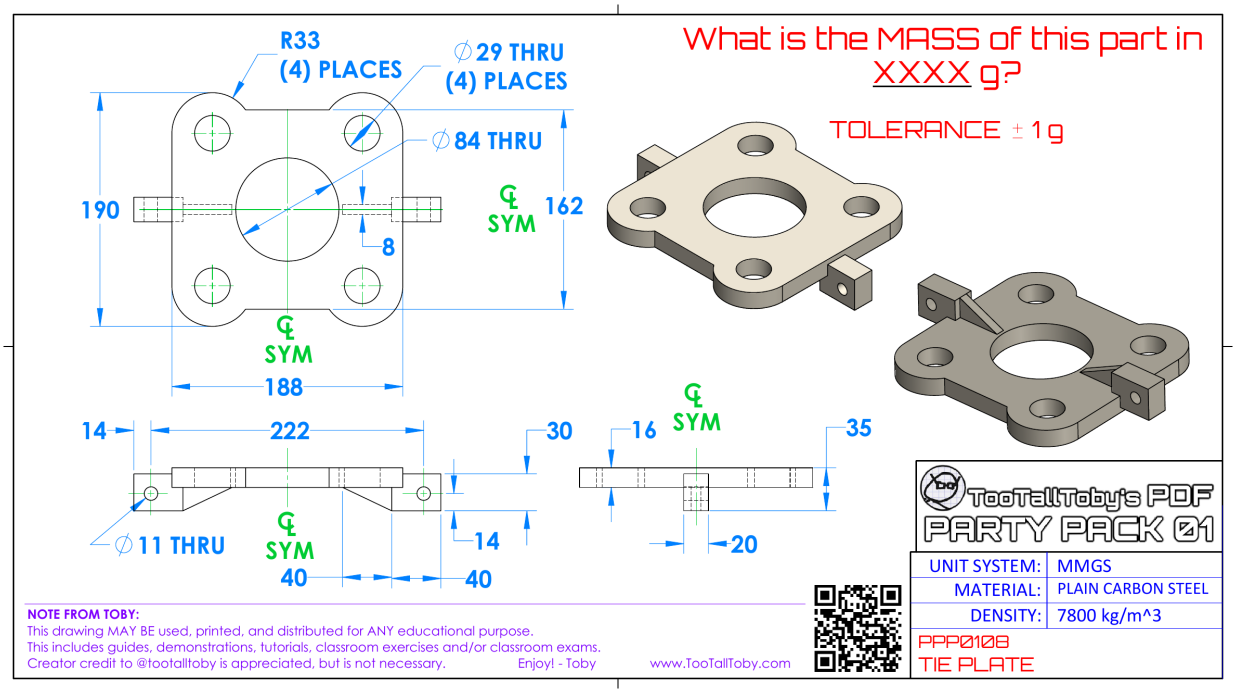
```
with BuildPart() as p:
    with BuildSketch() as s:
        Circle(130 / 2)
    extrude(amount=8)
    with BuildSketch(Plane.XY.offset(8)) as s2:
        Circle(84 / 2)
    extrude(amount=25 - 8)
    with BuildSketch(Plane.XY.offset(25)) as s3:
        Circle(35 / 2)
    extrude(amount=52 - 25)
    with BuildSketch() as s4:
        Circle(73 / 2)
    extrude(amount=18, mode=Mode.SUBTRACT)
    pln2 = p.part.faces().sort_by(Axis.Z)[5]
    with BuildSketch(Plane.XY.offset(52)) as s5:
        Circle(20 / 2)
    extrude(amount=-52, mode=Mode.SUBTRACT)
    fillet(
        p.part.edges()
        .filter_by(GeomType.CIRCLE)
        .sort_by(Axis.Z)[2:-2]
        .sort_by(SortBy.RADIUS)[1:],
        3,
    )
    pln = Plane(pln2)
    pln.origin = pln.origin + Vector(20 / 2, 0, 0)
    pln = pln.rotated((0, 45, 0))
    pln = pln.offset(-25 + 3 + 0.10)
    with BuildSketch(pln) as s6:
        Rectangle((73 - 35) / 2 * 1.414 + 5, 3)
    zz = extrude(amount=15, taper=-20 / 2, mode=Mode.PRIVATE)
    zz2 = split(zz, bisect_by=Plane.XY.offset(25), mode=Mode.PRIVATE)
    zz3 = split(zz2, bisect_by=Plane.YZ.offset(35 / 2 - 1), mode=Mode.PRIVATE)
    with PolarLocations(0, 3):
        add(zz3)
    with Locations(Plane.XY.offset(8)):
        with PolarLocations(107.95 / 2, 6):
            CounterBoreHole(6 / 2, 13 / 2, 4)

show(p)


got_mass = p.part.volume*densb
want_mass = 372.99
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-08 Tie Plate



Object Mass

3387.06 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-08 Tie Plate
"""

from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS

with BuildPart() as p:
    with BuildSketch() as s1:
        Rectangle(188 / 2 - 33, 162, align=(Align.MIN, Align.CENTER))
        with Locations((188 / 2 - 33, 0)):
            SlotOverall(190, 33 * 2, rotation=90)
        mirror(about=Plane.YZ)
        with GridLocations(188 - 2 * 33, 190 - 2 * 33, 2, 2):
            Circle(29 / 2, mode=Mode.SUBTRACT)
        Circle(84 / 2, mode=Mode.SUBTRACT)
    extrude(amount=16)

    with BuildPart() as p2:
```

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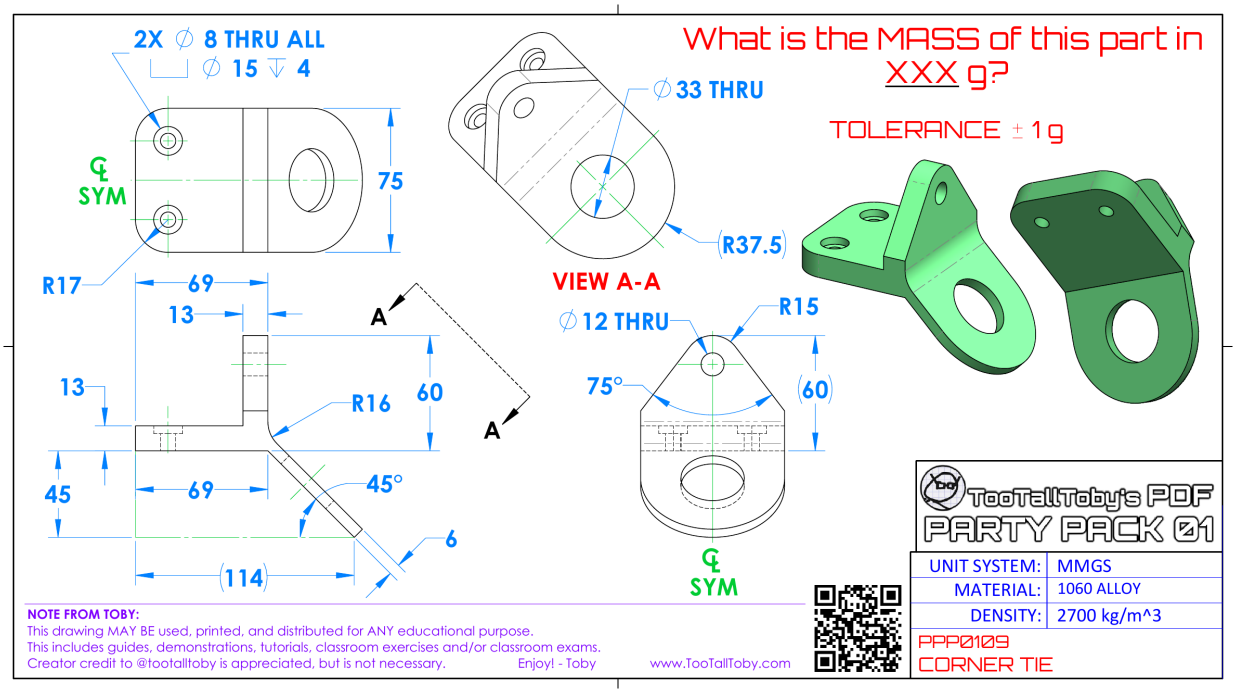
```
        with BuildSketch(Plane.XZ) as s2:
            with BuildLine() as l1:
                l1 = Polyline(
                    (222 / 2 + 14 - 40 - 40, 0),
                    (222 / 2 + 14 - 40, -35 + 16),
                    (222 / 2 + 14, -35 + 16),
                    (222 / 2 + 14, -35 + 16 + 30),
                    (222 / 2 + 14 - 40 - 40, -35 + 16 + 30),
                    close=True,
                )
            make_face()
            with Locations((222 / 2, -35 + 16 + 14)):
                Circle(11 / 2, mode=Mode.SUBTRACT)
        extrude(amount=20 / 2, both=True)
        with BuildSketch() as s3:
            with Locations(l1 @ 0):
                Rectangle(40 + 40, 8, align=(Align.MIN, Align.CENTER))
                with Locations((40, 0)):
                    Rectangle(40, 20, align=(Align.MIN, Align.CENTER))
        extrude(amount=30, both=True, mode=Mode.INTERSECT)
        mirror(about=Plane.YZ)

show(p)


got_mass = p.part.volume*densa
want_mass = 3387.06
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'
```

Party Pack 01-09 Corner Tie



Object Mass

307.23 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-09 Corner Tie
"""

from math import sqrt
from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS

with BuildPart() as ppp109:
    with BuildSketch() as one:
        Rectangle(69, 75, align=(Align.MAX, Align.CENTER))
        fillet(one.vertices().group_by(Axis.X)[0], 17)
    extrude(amount=13)
    centers = [
        arc.arc_center
        for arc in ppp109.edges().filter_by(GeomType.CIRCLE).group_by(Axis.Z)[-1]
    ]
    with Locations(*centers):
        CounterBoreHole(radius=8 / 2, counter_bore_radius=15 / 2, counter_bore_depth=4)
```

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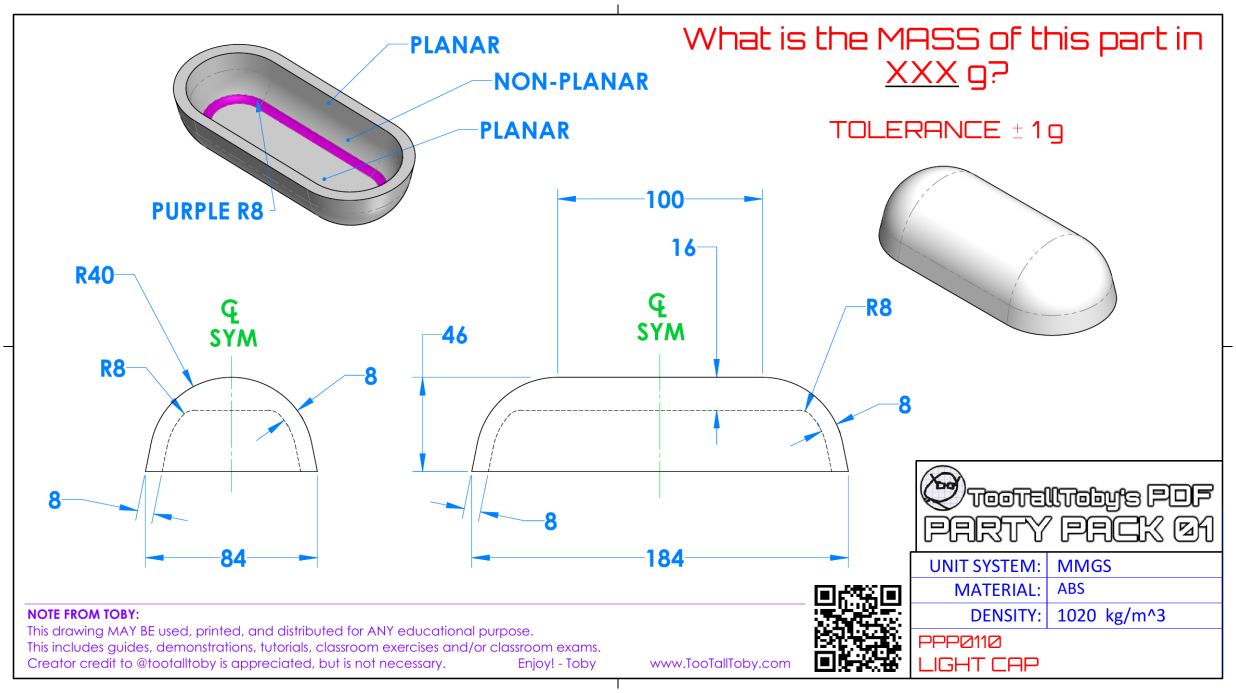
```
    with BuildSketch(Plane.YZ) as two:
        with Locations((0, 45)):
            Circle(15)
        with BuildLine() as bl:
            c = Line((75 / 2, 0), (75 / 2, 60), mode=Mode.PRIVATE)
            u = two.edge().find_tangent(75 / 2 + 90)[0]  # where is the slope 75/2?
            l1 = IntersectingLine(
                two.edge().position_at(u), -two.edge().tangent_at(u), other=c
            )
            Line(l1 @ 0, (0, 45))
            Polyline((0, 0), c @ 0, l1 @ 1)
            mirror(about=Plane.YZ)
        make_face()
        with Locations((0, 45)):
            Circle(12 / 2, mode=Mode.SUBTRACT)
    extrude(amount=-13)

    with BuildSketch(Plane((0, 0, 0), x_dir=(1, 0, 0), z_dir=(1, 0, 1))) as three:
        Rectangle(45 * 2 / sqrt(2) - 37.5, 75, align=(Align.MIN, Align.CENTER))
        with Locations(three.edges().sort_by(Axis.X)[-1].center()):
            Circle(37.5)
            Circle(33 / 2, mode=Mode.SUBTRACT)
        split(bisect_by=Plane.YZ)
    extrude(amount=6)
    f = ppp109.faces().filter_by(Axis((0, 0, 0), (-1, 0, 1)))[0]
    extrude(f, until=Until.NEXT)
    fillet(ppp109.edges().filter_by(Axis.Y).sort_by(Axis.Z)[2], 16)
    # extrude(f, amount=10)
    # fillet(ppp109.edges(Select.NEW), 16)


show(ppp109)

got_mass = ppp109.part.volume * densb
want_mass = 307.23
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.2f} g")
assert delta < tolerance, f"{got_mass=}, {want_mass=}, {delta=}, {tolerance=}"
```

Party Pack 01-10 Light Cap



Object Mass

211.30 g

Reference Implementation

```
"""
Too Tall Toby Party Pack 01-10 Light Cap
"""

from math import sqrt, asin, pi
from build123d import *
from ocp_vscode import *

densa = 7800 / 1e6  # carbon steel density g/mm^3
densb = 2700 / 1e6  # aluminum alloy
densc = 1020 / 1e6  # ABS

# The smaller cross-section is defined as having R40, height 46,
# and base width 84, so clearly it's not entirely a half-circle or
# similar; the base's extreme points need to connect via tangents
# to the R40 arc centered 6mm above the baseline.
#
# Compute the angle of the tangent line (working with the
# left/negativeX side, given symmetry) by observing the tangent
# point (T), the circle's center (O), and the baseline's edge (P)
# form a right triangle, so:

OT=40
```

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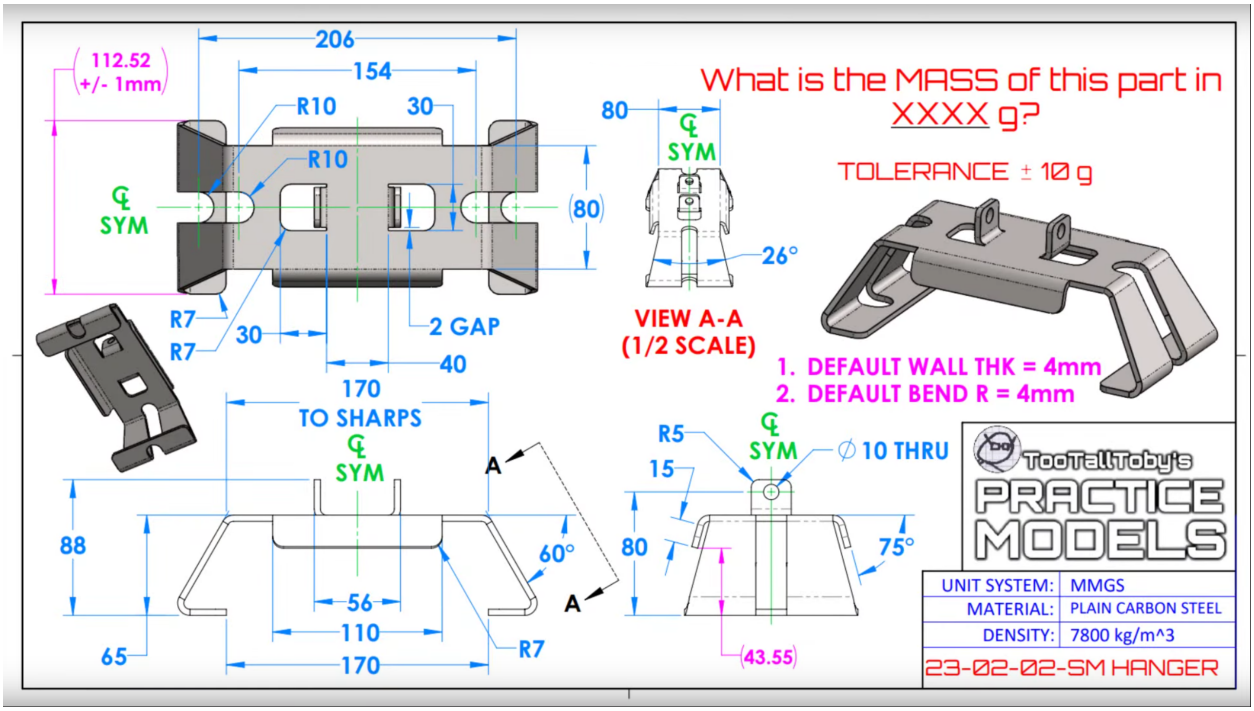
```
OP=sqrt((-84/2)**2+(-6)**2)
TP=sqrt(OP**2-40**2)
OPT_degrees = asin(OT/OP) * 180/pi
# Correct for the fact that OP isn't horizontal.
OP_to_X_axis_degrees = asin(6/OP) * 180/pi
left_tangent_degrees = OPT_degrees + OP_to_X_axis_degrees
left_tangent_length = TP
with BuildPart() as outer:
    with BuildSketch(Plane.XZ) as sk:
        with BuildLine():
            l1 = PolarLine(start=(-84/2, 0), length=left_tangent_length, angle=left_
↪tangent_degrees)
            l2 = TangentArc(l1@1, (0, 46), tangent=l1%1)
            l3 = offset(amount=-8, side=Side.RIGHT, closed=False, mode=Mode.ADD)
            l4 = Line(l1@0, l3@1)
            l5 = Line(l3@0, l2@1)
        make_face()

        with BuildLine():
            l6 = Line(l2 @ 1, (0, 46 - 16))
            l7 = IntersectingLine(start=l6 @ 1, direction=(-1, 0), other=l3)
            l8 = TangentArc(l7 @ 1, l2 @ 1, tangent=(-1, 0), tangent_from_first=False)

        make_face()

    revolve(axis=Axis.Z)
sk = sk.sketch & Plane.XZ*Rectangle(1000, 1000, align=[Align.CENTER, Align.MIN])
positive_Z = Box(100, 100, 100, align=[Align.CENTER, Align.MIN, Align.MIN])
p = outer.part & positive_Z
cross_section = sk + mirror(sk, about=Plane.YZ)
p += extrude(cross_section, amount=50)
p += mirror(p, about=Plane.XZ.offset(50))
p += fillet(p.edges().filter_by(GeomType.LINE).filter_by(Axis.Y).group_by(Axis.Z)[-1],␣
↪radius=8)
ppp0110 = p

got_mass = ppp0110.volume*densc
want_mass = 211.30
tolerance = 1
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.1f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'

show(ppp0110)
```

23-02-02 SM Hanger



Object Mass

1028g +/- 10g

Reference Implementation

```
"""
Creation of a complex sheet metal part

name: ttt_sm_hanger.py
by:   Gumyr
date: July 17, 2023

desc:
    This example implements the sheet metal part described in Too Tall Toby's
    sm_hanger CAD challenge.

    Notably, a BuildLine/Curve object is filleted by providing all the vertices
    and allowing the fillet operation filter out the end vertices. The
    make_brake_formed operation is used both in Algebra and Builder mode to
    create a sheet metal part from just an outline and some dimensions.
    license:

    Copyright 2023 Gumyr

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    You may obtain a copy of the License at
```

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```
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    limitations under the License.

"""

from build123d import *
from ocp_vscode import *

sheet_thickness = 4 * MM

# Create the main body from a side profile
with BuildPart() as side:
    d = Vector(1, 0, 0).rotate(Axis.Y, 60)
    with BuildLine(Plane.XZ) as side_line:
        l1 = Line((0, 65), (170 / 2, 65))
        l2 = PolarLine(l1 @ 1, length=65, direction=d, length_mode=LengthMode.VERTICAL)
        l3 = Line(l2 @ 1, (170 / 2, 0))
        fillet(side_line.vertices(), 7)
    make_brake_formed(
        thickness=sheet_thickness,
        station_widths=[40, 40, 40, 112.52 / 2, 112.52 / 2, 112.52 / 2],
        side=Side.RIGHT,
    )
    fe = side.edges().filter_by(Axis.Z).group_by(Axis.Z)[0].sort_by(Axis.Y)[-1]
    fillet(fe, radius=7)

# Create the "wings" at the top
with BuildPart() as wing:
    with BuildLine(Plane.YZ) as wing_line:
        l1 = Line((0, 65), (80 / 2 + 1.526 * sheet_thickness, 65))
        PolarLine(l1 @ 1, 20.371288916, direction=Vector(0, 1, 0).rotate(Axis.X, -75))
        fillet(wing_line.vertices(), 7)
    make_brake_formed(
        thickness=sheet_thickness,
        station_widths=110 / 2,
        side=Side.RIGHT,
    )
    bottom_edge = wing.edges().group_by(Axis.X)[-1].sort_by(Axis.Z)[0]
    fillet(bottom_edge, radius=7)

# Create the tab at the top in Algebra mode
tab_line = Plane.XZ * Polyline(
    (20, 65 - sheet_thickness), (56 / 2, 65 - sheet_thickness), (56 / 2, 88)
)
tab_line = fillet(tab_line.vertices(), 7)
tab = make_brake_formed(sheet_thickness, 8, tab_line, Side.RIGHT)
tab = fillet(tab.edges().filter_by(Axis.X).group_by(Axis.Z)[-1].sort_by(Axis.Y)[-1], 5)
```

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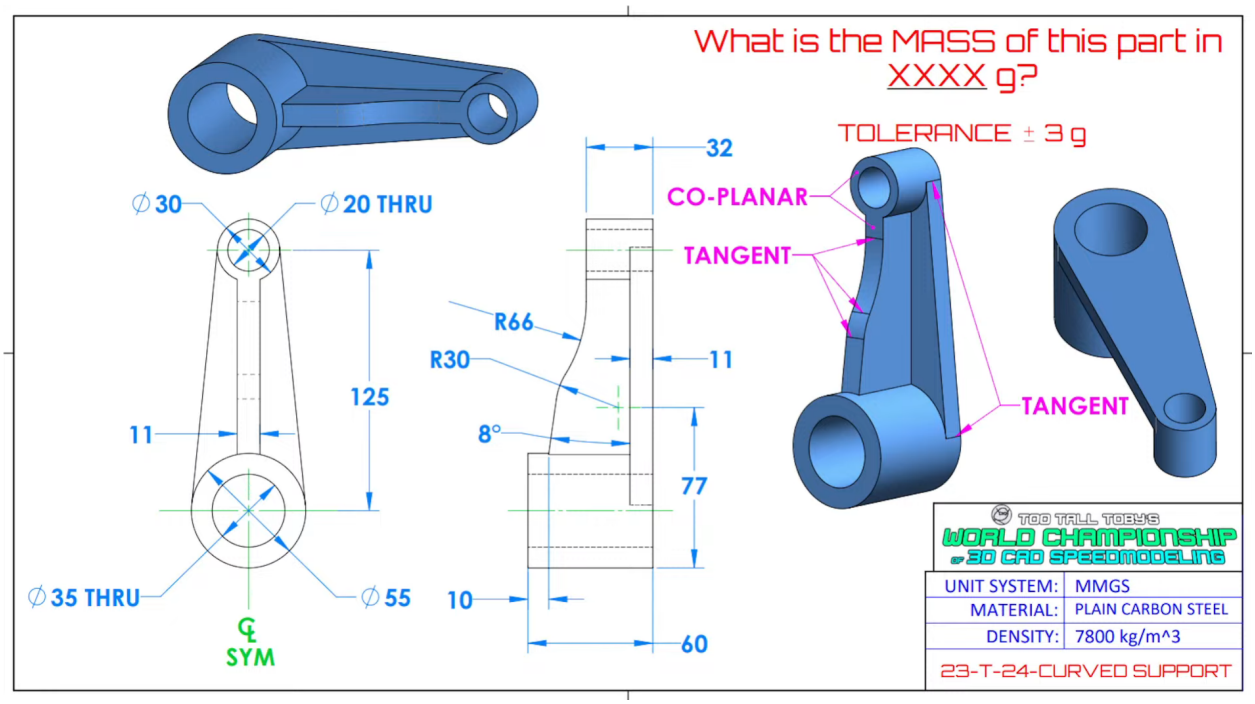
```
tab -= Pos((0, 0, 80)) * Rot(0, 90, 0) * Hole(5, 100)

# Combine the parts together
with BuildPart() as sm_hanger:
    add([side.part, wing.part])
    mirror(about=Plane.XZ)
    with BuildSketch(Plane.XY.offset(65)) as h1:
        with Locations((20, 0)):
            Rectangle(30, 30, align=(Align.MIN, Align.CENTER))
            fillet(h1.vertices().group_by(Axis.X)[-1], 7)
        SlotCenterPoint((154, 0), (154 / 2, 0), 20)
    extrude(amount=-40, mode=Mode.SUBTRACT)
    with BuildSketch() as h2:
        SlotCenterPoint((206, 0), (206 / 2, 0), 20)
    extrude(amount=40, mode=Mode.SUBTRACT)
    add(tab)
    mirror(about=Plane.YZ)
    mirror(about=Plane.XZ)

got_mass = sm_hanger.part.volume * 7800 * 1e-6
want_mass = 1028
tolerance = 10
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.1f} g")
# assert delta < tolerance, f"{got_mass=}, {want_mass=}, {delta=}, {tolerance=}"

# assert abs(got_mass - 1028) < 10, f"{got_mass=}, want=1028, tolerance=10"

show(sm_hanger)
```

23-T-24 Curved Support



Object Mass

1294 g

Reference Implementation

```
"""
Too Tall Toby challenge 23-T-24 CURVED SUPPORT
"""

from math import sin, cos, tan, radians
from build123d import *
from ocp_vscode import *
import sympy

# This problem uses the sympy symbolic math solver

# Define the symbols for the unknowns
# - the center of the radius 30 arc (x30, y30)
# - the center of the radius 66 arc (x66, y66)
# - end of the 8° line (l8x, l8y)
# - the point with the radius 30 and 66 arc meet i30_66
# - the start of the horizontal line lh
y30, x66, xl8, yl8 = sympy.symbols("y30 x66 xl8 yl8")
x30 = 77 - 55 / 2
y66 = 66 + 32

# There are 4 unknowns so we need 4 equations
equations = [
```

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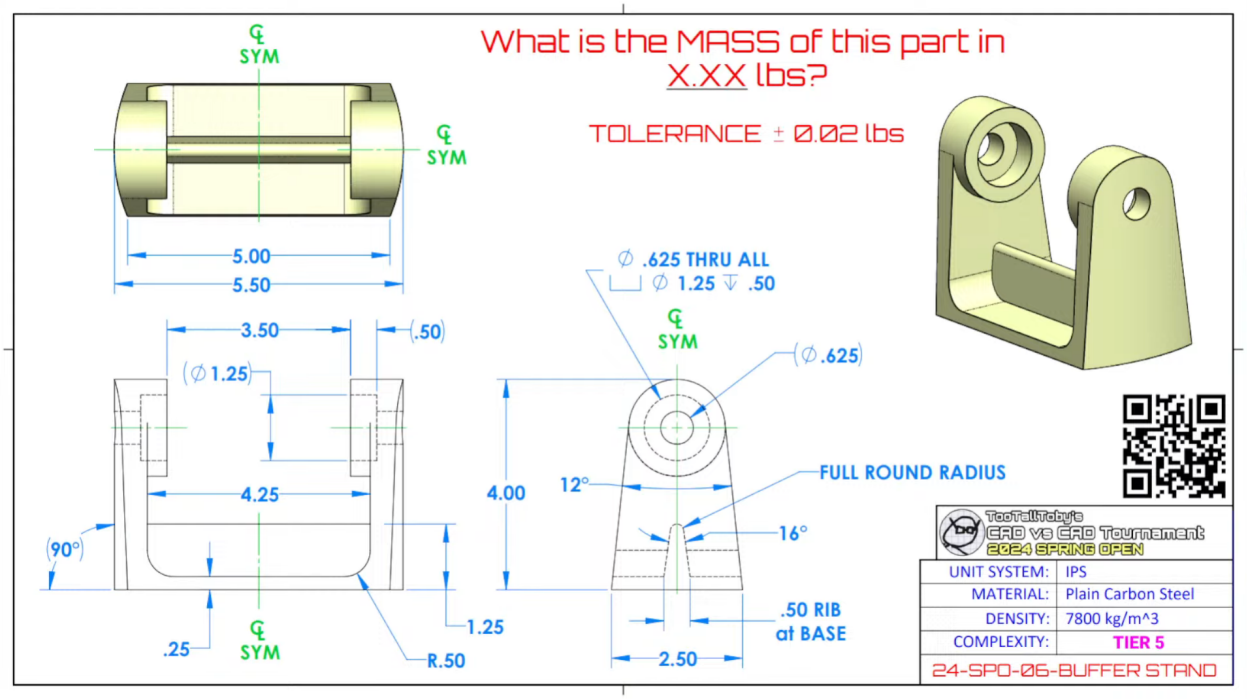
```
    (x66 - x30) ** 2 + (y66 - y30) ** 2 - (66 + 30) ** 2,  # distance between centers
    xl8 - (x30 + 30 * sin(radians(8))),  # 8 degree slope
    yl8 - (y30 + 30 * cos(radians(8))),  # 8 degree slope
    (yl8 - 50) / (55 / 2 - xl8) - tan(radians(8)),  # 8 degree slope
]
# There are two solutions but we want the 2nd one
solution = {k: float(v) for k,v in sympy.solve(equations, dict=True)[1].items()}

# Create the critical points
c30 = Vector(x30, solution[y30])
c66 = Vector(solution[x66], y66)
l8 = Vector(solution[xl8], solution[yl8])
i30_66 = Line(c30, c66) @ (30 / (30 + 66))
lh = Vector(c66.X, 32)

with BuildLine() as profile:
    l1 = Line((55 / 2, 50), l8)
    l2 = RadiusArc(l1 @ 1, i30_66, 30)
    l3 = RadiusArc(l2 @ 1, lh, -66)
    l4 = Polyline(l3 @ 1, (125, 32), (125, 0), (0, 0), (0, (l1 @ 0).Y), l1 @ 0)

with BuildPart() as curved_support:
    with BuildSketch() as base_plan:
        c_8_degrees = Circle(55 / 2)
        with Locations((0, 125)):
            Circle(30 / 2)
        base_hull = make_hull(mode=Mode.PRIVATE)
    extrude(amount=32)
    extrude(c_8_degrees, amount=60)
    extrude(base_hull, amount=11)
    with BuildSketch(Plane.YZ) as bridge:
        make_face(profile.edges())
    extrude(amount=11 / 2, both=True)
    Hole(35 / 2)
    with Locations((0, 125)):
        Hole(20 / 2)

got_mass = curved_support.part.volume * 7800e-6
want_mass = 1294
delta = abs(got_mass - want_mass)
tolerance = 3
print(f"Mass: {got_mass:0.1f} g")
assert delta < tolerance, f'{got_mass=}, {want_mass=}, {delta=}, {tolerance=}'

show(curved_support)
```

24-SPO-06 Buffer Stand



Object Mass

3.92 lbs

Reference Implementation

```
from build123d import *
from ocp_vscode import show

with BuildPart() as p:
    with BuildSketch() as xy:
        with BuildLine():
            l1 = ThreePointArc((5 / 2, -1.25), (5.5 / 2, 0), (5 / 2, 1.25))
            Polyline(l1 @ 0, (0, -1.25), (0, 1.25), l1 @ 1)
        make_face()
    extrude(amount=4)

    with BuildSketch(Plane.YZ) as yz:
        Trapezoid(2.5, 4, 90 - 6, align=(Align.CENTER, Align.MIN))
        full_round(yz.edges().sort_by(SortBy.LENGTH)[0])
        circle_edge = yz.edges().filter_by(GeomType.CIRCLE)[0]
        arc_center = circle_edge.arc_center
        arc_radius = circle_edge.radius
    extrude(amount=10, mode=Mode.INTERSECT)

    # To avoid OCCT problems, don't attempt to extend the top arc, remove instead
    with BuildPart(mode=Mode.SUBTRACT) as internals:
        y = p.edges().filter_by(Axis.X).sort_by(Axis.Z)[-1].center().Z
```

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```
        with BuildSketch(Plane.YZ.offset(4.25 / 2)) as yz:
            Trapezoid(2.5, y, 90 - 6, align=(Align.CENTER, Align.MIN))
            with Locations(arc_center):
                Circle(arc_radius, mode=Mode.SUBTRACT)
        extrude(amount=-(4.25 - 3.5) / 2)

        with BuildSketch(Plane.YZ.offset(3.5 / 2)) as yz:
            Trapezoid(2.5, 4, 90 - 6, align=(Align.CENTER, Align.MIN))
        extrude(amount=-3.5 / 2)

        with BuildSketch(Plane.XZ.offset(-2)) as xz:
            with Locations((0, 4)):
                RectangleRounded(4.25, 7.5, 0.5)
        extrude(amount=4, mode=Mode.INTERSECT)

    with Locations(p.faces(Select.LAST).filter_by(GeomType.PLANE).sort_by(Axis.Z)[-1]):
        CounterBoreHole(0.625 / 2, 1.25 / 2, 0.5)

    with BuildSketch(Plane.YZ) as rib:
        with Locations((0, 0.25)):
            Trapezoid(0.5, 1, 90 - 8, align=(Align.CENTER, Align.MIN))
        full_round(rib.edges().sort_by(SortBy.LENGTH)[0])
    extrude(amount=4.25 / 2)

    mirror(about=Plane.YZ)

part = scale(p.part, IN)


got_mass = part.volume * 7800e-6 / LB
want_mass = 3.923
tolerance = 0.02
delta = abs(got_mass - want_mass)
print(f"Mass: {got_mass:0.1f} lbs")
assert delta < tolerance, f"{got_mass=}, {want_mass=}, {delta=}, {tolerance=}"

show(p)
```

1.9.8 Tutorial: Reconstructing a Design from an STL

This tutorial describes a practical workflow for using *detect_primitives()* to help reconstruct a parametric build123d model from an STL mesh.

This is not a push-button STL-to-CAD converter. It is a mesh-guided redesign process. The goal is to extract enough analytic structure from a triangulated model to make manual reconstruction faster and more reliable.

> **Warning**
>
> Rebuilding a design from STL is usually slow, approximate, and manual. STL files contain triangles, not modeling intent. Even when *detect_primitives()* finds useful planes, cylinders, and spheres, the final build123d model still needs to be designed deliberately.

Before working through this tutorial, review Steps 1-3 of *Designing a Part in build123d*. The same ideas apply here:

- identify planes of symmetry
- identify likely axes of rotation
- choose a convenient origin before doing any serious work

These preparation steps often reduce the amount of mesh that needs to be reconstructed to one half, one quarter, or even less.

Overview

The workflow described here is:

1. Import the STL with *Mesher*.
2. Split the mesh by symmetry planes and isolate the region to redesign.
3. Save that reduced mesh section as a BREP file.
4. Reload the BREP section while iterating on reconstruction.
5. Run *detect_primitives()*.
6. Inspect the returned primitives, leftovers, and generated code.
7. Rebuild the design intentionally from those clues.

The key output of `detect_primitives` is guidance:

- `primitives` shows what was recognized analytically
- `leftovers` shows what was not covered
- `code_lines` provides algebra-mode fragments that often reveal common planes and likely sketch structure

Why Cache a Working Section as BREP?

Importing STL with *Mesher* is convenient, but large meshes can be slow to load and process. Once a useful section of the part has been isolated, save it as a BREP file and use that for repeated experimentation.

BREP files reload much more efficiently in build123d and are better suited to an iterative reconstruction script.

Preparing the Mesh

Start with the STL import and isolate the smallest useful section of the part.

```
from build123d import *

importer = Mesher()
full_mesh = importer.read("target_part.stl")[0]

# Example: reduce the work to one quarter of a symmetric model
quarter_mesh = split(full_mesh, Plane.YZ)
quarter_mesh = split(quarter_mesh, Plane.XZ)

export_brep(quarter_mesh, "target_part_quarter.brep")
```

The exact planes depend on the part. The point is not to begin running `detect_primitives` on the full mesh if symmetry can remove most of the work.

Reconstruction Script

Once the mesh section has been cached as BREP, iterate on a separate script or enable a reconstruction section of the same script with a Boolean switch.

```
from build123d import *

working_mesh = import_brep("target_part_quarter.brep")

primitives, leftovers, code_lines = detect_primitives(working_mesh)

print(*code_lines, sep="\n")
```

This call returns three complementary outputs:

primitives
: A *ShapeList* of analytic faces that were recognized from the mesh. These are typically planes, cylinders, and spheres. Planar primitives are returned as rectangles sized to the planar region's bounding box, not as the original tessellated mesh patch.

leftovers
: Mesh faces that were not matched by the primitive detectors. These indicate freeform regions, noisy regions, or places where manual work is still required.

code_lines
: Generated algebra-mode code corresponding to the recognized primitives.

Inspecting the Results

The inspection step is the heart of this workflow.

1. Examine the primitives visually

Look at the returned `primitives` and decide what the detector found well.

Useful questions include:

- Are the expected planar faces present?
- Do fillets appear as cylinders?
- Do rounded corners appear as spheres?
- Are repeated features recognized consistently?

In a simple mechanical part, good output often consists of a small number of common planes, repeated cylinders with similar radii, and only a few leftovers.

2. Examine the leftovers

`leftovers` show what still needs manual interpretation.

Large leftover regions often indicate one of three things:

- the part contains geometry that is not well approximated by planes, cylinders, or spheres
- the mesh is noisy or irregular
- the working section is still too large to interpret comfortably

If too much of the mesh appears in `leftovers`, it may be better to refine the working section, identify more symmetry, or redesign that area manually instead of trying to automate it further.

3. Examine the generated code

The generated `code_lines` are intentionally written in Algebra mode and use `Plane * Pos` structure to make repeated placement patterns easier to spot.

This often helps answer questions such as:

- which faces lie on the same construction plane?
- which circles belong to the same sketch?
- which cylindrical or spherical regions are repeated instances of one feature?

Treat this code as an annotated report, not necessarily as the final model.

For planar parts in particular, the generated lines are often naturally grouped by plane. A sequence such as `Plane.XY.offset(...)` with a few repeated offset values usually indicates related structure that may belong to one sketch or one construction stage.

Worked Example: Filleted Box

As a controlled example, consider a filleted box:

```
fillet_box = fillet(Box(1, 1, 1).edges(), 0.1)
```

Running `detect_primitives` on this geometry produces output like:

```
r00 = Plane.XY.offset(-0.5) * Pos(-0.4, -0.4) * Rectangle(0.8, 0.8, align=Align.MIN)
c01 = Plane.XY.offset(-0.4) * Pos(0.4, 0.4) * Face.extrude(Circle(0.0999996).edge(), (0,␣
↪0, 0.8))
c02 = Plane.XY.offset(-0.4) * Pos(-0.4, 0.4) * Face.extrude(Circle(0.0999996).edge(), (0,
↪ 0, 0.8))
c03 = Plane.XY.offset(-0.4) * Pos(-0.4, -0.4) * Face.extrude(Circle(0.0999996).edge(),␣
↪(0, 0, 0.8))
c04 = Plane.XY.offset(-0.4) * Pos(0.4, -0.4) * Face.extrude(Circle(0.0999996).edge(), (0,
↪ 0, 0.8))
r05 = Plane.XY.offset(0.5) * Pos(-0.4, -0.4) * Rectangle(0.8, 0.8, align=Align.MIN)
r06 = Plane.YZ.offset(-0.5) * Pos(-0.4, -0.4) * Rectangle(0.8, 0.8, align=Align.MIN)
c07 = Plane.YZ.offset(-0.4) * Pos(-0.4, -0.4) * Face.extrude(Circle(0.0999996).edge(),␣
↪(0, 0, 0.8))
c08 = Plane.YZ.offset(-0.4) * Pos(-0.4, 0.4) * Face.extrude(Circle(0.0999996).edge(), (0,
↪ 0, 0.8))
c09 = Plane.YZ.offset(-0.4) * Pos(0.4, -0.4) * Face.extrude(Circle(0.0999996).edge(), (0,
↪ 0, 0.8))
c10 = Plane.YZ.offset(-0.4) * Pos(0.4, 0.4) * Face.extrude(Circle(0.0999996).edge(), (0,␣
↪0, 0.8))
r11 = Plane.YZ.offset(0.5) * Pos(-0.4, -0.4) * Rectangle(0.8, 0.8, align=Align.MIN)
r12 = Plane.ZX.offset(-0.5) * Pos(-0.4, -0.4) * Rectangle(0.8, 0.8, align=Align.MIN)
c13 = Plane.ZX.offset(-0.4) * Pos(-0.4, 0.4) * Face.extrude(Circle(0.0999996).edge(), (0,
↪ 0, 0.8))
c14 = Plane.ZX.offset(-0.4) * Pos(-0.4, -0.4) * Face.extrude(Circle(0.0999996).edge(),␣
↪(0, 0, 0.8))
c15 = Plane.ZX.offset(-0.4) * Pos(0.4, -0.4) * Face.extrude(Circle(0.0999996).edge(), (0,
↪ 0, 0.8))
c16 = Plane.ZX.offset(-0.4) * Pos(0.4, 0.4) * Face.extrude(Circle(0.0999996).edge(), (0,␣
↪0, 0.8))
r17 = Plane.ZX.offset(0.5) * Pos(-0.4, -0.4) * Rectangle(0.8, 0.8, align=Align.MIN)
```

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```
s18 = Pos((0.399999, -0.399999, 0.400026)) * Sphere(0.099983).faces().filter_by(GeomType.
↪SPHERE)[0]
s19 = Pos((-0.399999, 0.399999, -0.400026)) * Sphere(0.099983).faces().filter_
↪by(GeomType.SPHERE)[0]
s20 = Pos((-0.399999, -0.399999, -0.400026)) * Sphere(0.099983).faces().filter_
↪by(GeomType.SPHERE)[0]
s21 = Pos((0.399999, 0.399999, -0.400026)) * Sphere(0.099983).faces().filter_by(GeomType.
↪SPHERE)[0]
s22 = Pos((-0.399999, 0.400026, 0.399999)) * Sphere(0.099983).faces().filter_by(GeomType.
↪SPHERE)[0]
s23 = Pos((-0.399999, -0.399999, 0.400026)) * Sphere(0.099983).faces().filter_
↪by(GeomType.SPHERE)[0]
s24 = Pos((0.399999, 0.400026, 0.399999)) * Sphere(0.099983).faces().filter_by(GeomType.
↪SPHERE)[0]
s25 = Pos((0.399999, -0.399999, -0.400026)) * Sphere(0.099983).faces().filter_
↪by(GeomType.SPHERE)[0]
```

This output is informative in several ways:

- the six box faces appear as rectangles on three principal planes
- the edge fillets appear as cylinders grouped around those same planes
- the corner blends appear as spheres near the eight cube corners

The generated code is also structured by plane:

- `Plane.XY.offset(...)` appears with three distinct offsets
- `Plane.YZ.offset(...)` appears with three distinct offsets
- `Plane.ZX.offset(...)` appears with three distinct offsets

That organization is often more useful than any one primitive by itself because it suggests how the model could be regrouped into sketches and construction steps.

Although this output is correct and useful, it still does not represent the best final build123d model. The original design intent is much simpler:

```
fillet(Box(1, 1, 1).edges(), 0.1)
```

That is a good example of the main lesson of this tutorial: the generated code helps reveal structure, but the final model should usually be rewritten in a cleaner, higher-level form.

Turning Primitive Hints into Sketches

Once repeated planes become obvious in `code_lines`, start grouping related features into sketches and features of your own.

For example:

- several rectangles on `Plane.XY` may indicate one base sketch and one or more extrusions
- repeated circles on one plane may indicate hole or boss locations
- a collection of cylinders with the same radius may indicate that a fillet or round was part of the original design intent

The generated code is often most useful when treated as:

- a list of candidate construction planes

- a list of likely sketch elements
- a list of repeated primitive sizes and placements

Signs of Good Output

`detect_primitives` is most helpful when:

- the mesh is reasonably clean
- the part is mostly mechanical
- many surfaces are planar, cylindrical, or spherical
- there are clear planes of symmetry or repeated features

In these cases, primitives and generated code often cluster into obvious reconstruction steps.

Signs of Poor Output

Expect more manual work when:

- the mesh contains freeform surfaces
- the STL is noisy or heavily tessellated
- the part has no obvious symmetry
- many important regions remain in `leftovers`
- the generated code contains many tiny or redundant fragments

When this happens, it may be faster to use the mesh only as a visual reference and rebuild the part manually from dimensions and intent.

Summary

The STL reconstruction workflow in build123d is:

1. analyze the part as a designer, not as a mesh processor
2. isolate the smallest useful region with symmetry and splitting
3. cache that region as BREP
4. run *detect_primitives()*
5. inspect primitives, leftovers, and code
6. rebuild the part intentionally in build123d

The most useful mindset is to treat `detect_primitives` as a design assistant. It can show where the planes, cylinders, and spheres probably are, but the final parametric model still comes from careful human interpretation.

1.9.9 Surface Modeling

Surface modeling refers to the direct creation and manipulation of the skin of a 3D object—its bounding faces—rather than starting from volumetric primitives or solid operations.

Instead of defining a shape by extruding or revolving a 2D profile to fill a volume, surface modeling focuses on building the individual curved or planar faces that together define the outer boundary of a part. This approach allows for precise control of complex freeform geometry such as aerodynamic surfaces, boat hulls, or organic transitions that cannot easily be expressed with simple parametric solids.

In build123d, as in other CAD kernels based on BREP (Boundary Representation) modeling, all solids are ultimately defined by their boundaries: a hierarchy of faces, edges, and vertices. Each face represents a finite patch of a geometric

surface (plane, cylinder, Bézier patch, etc.) bounded by one or more edge loops or wires. When adjacent faces share edges consistently and close into a continuous boundary, they form a manifold *Shell*—the watertight surface of a volume. If this shell is properly oriented and encloses a finite region of space, the model becomes a solid.

Surface modeling therefore operates at the most fundamental level of BREP construction. Rather than relying on higher-level modeling operations to implicitly generate faces, it allows you to construct and connect those faces explicitly. This provides a path to build geometry that blends analytical and freeform shapes seamlessly, with full control over continuity, tangency, and curvature across boundaries.

This section provides: - A concise overview of surface-building tools in build123d - Hands-on tutorials, from fundamentals to advanced techniques like Gordon surfaces

Available surface methods

Methods on *Face* for creating non-planar surfaces:

- *make_bezier_surface()*
- *make_gordon_surface()*
- *make_surface()*
- *make_surface_from_array_of_points()*
- *make_surface_from_curves()*
- *make_surface_patch()*

Note

Surface modeling is an advanced technique. Robust results usually come from reusing the same *Edge* objects across adjacent faces and ensuring the final *Shell* is *water-tight* or *manifold* (no gaps).

Tutorial: Heart Token (Basics)

This hands-on tutorial introduces the fundamentals of surface modeling by building a heart-shaped token from a small set of non-planar faces. We'll create non-planar surfaces, mirror them, add side faces, and assemble a closed shell into a solid.

As described in the *topology_* section, a BREP model consists of vertices, edges, faces, and other elements that define the boundary of an object. When creating objects with non-planar faces, it is often more convenient to explicitly create the boundary faces of the object. To illustrate this process, we will create the following game token:

Useful *Face* creation methods include *make_surface()*, *make_bezier_surface()*, and *make_surface_from_array_of_points()*. See the *Surface Modeling* overview for the full list.

In this case, we'll use the `make_surface` method, providing it with the edges that define the perimeter of the surface and a central point on that surface.

To create the perimeter, we'll define the perimeter edges. Since the heart is symmetric, we'll only create half of its surface here:

```
from build123d import *
from ocp_vscode import show

# Create the edges of one half the heart surface
l1 = JernArc((0, 0), (1, 1.4), 40, -17)
l2 = JernArc(l1 @ 1, l1 % 1, 4.5, 175)
l3 = IntersectingLine(l2 @ 1, l2 % 1, other=Edge.make_line((0, 0), (0, 20)))
```

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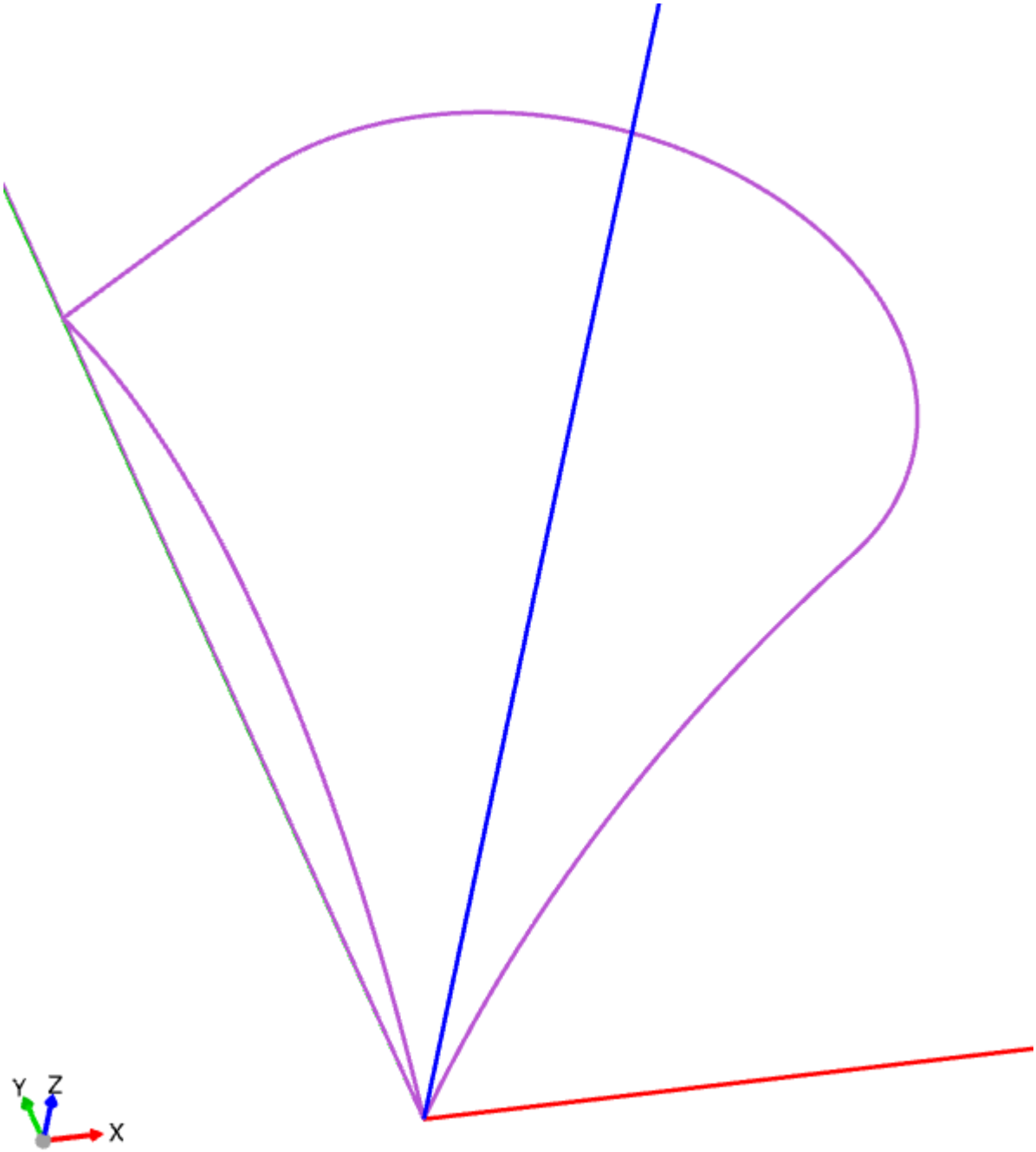
```
l4 = ThreePointArc(l3 @ 1, (0, 0, 1.5) + (l3 @ 1 + l1 @ 0) / 2, l1 @ 0)
heart_half = Wire([l1, l2, l3, l4])
```

Note that `l4` is not in the same plane as the other lines; it defines the center line of the heart and archs up off `Plane.XY`.

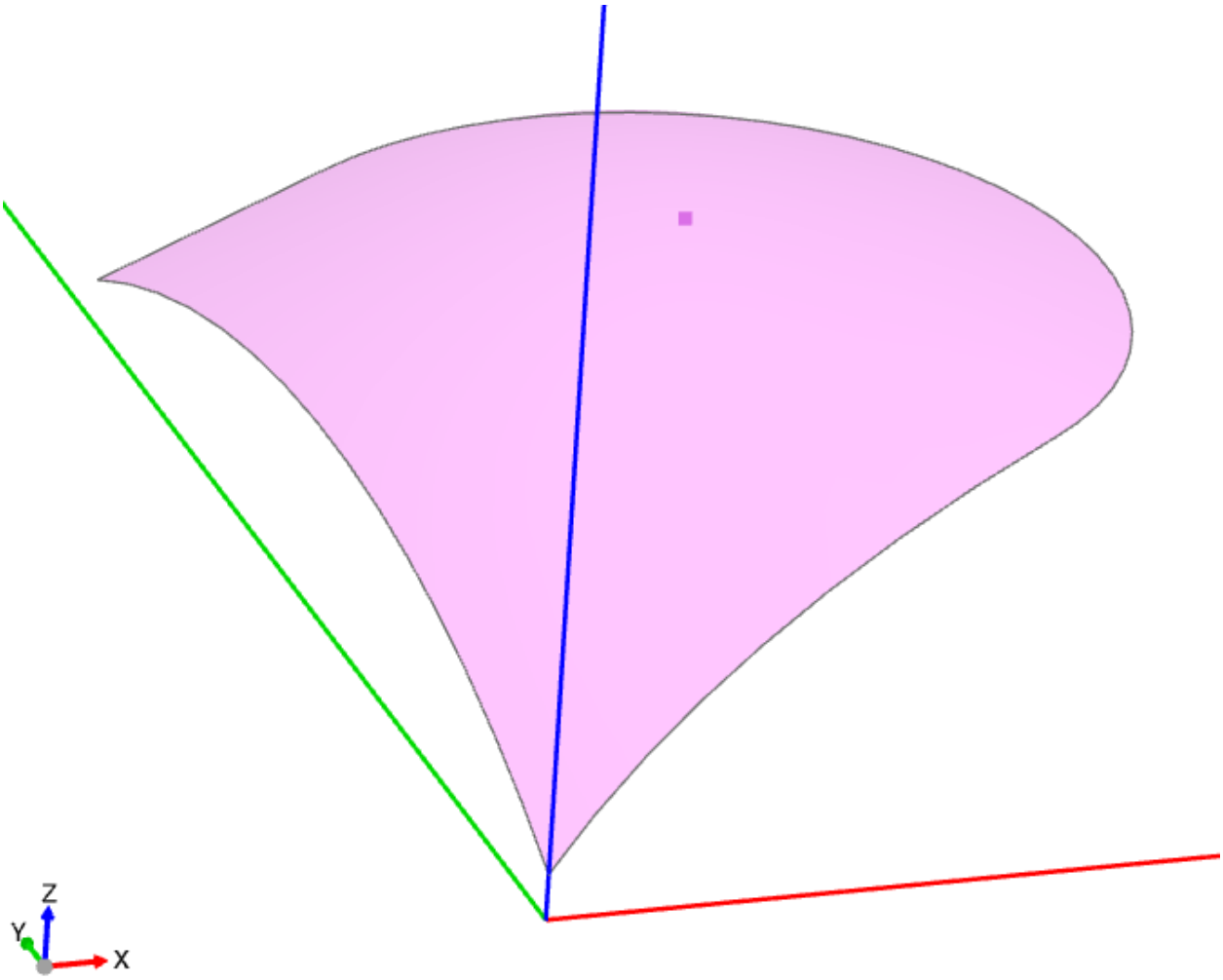


In preparation for creating the surface, we'll define a point on the surface:

```
# Create a point elevated off the center
surface_pnt = l2.arc_center + (0, 0, 1.5)
```

We will then use this point to create a non-planar Face:

```
# Create the surface from the edges and point
top_right_surface = Pos(Z=0.5) * -Face.make_surface(heart_half, [surface_pnt])
```



Note that the surface was raised up by 0.5 using an Algebra expression with Pos. Also, note that the - in front of Face simply flips the face normal so that the colored side is up, which isn't necessary but helps with viewing.

Now that one half of the top of the heart has been created, the remainder of the top and bottom can be created by mirroring:

```
# Use the mirror method to create the other top and bottom surfaces
top_left_surface = top_right_surface.mirror(Plane.YZ)
bottom_right_surface = top_right_surface.mirror(Plane.XY)
bottom_left_surface = -top_left_surface.mirror(Plane.XY)
```

The sides of the heart are going to be created by extruding the outside of the perimeter as follows:

```
# Create the left and right sides
left_wire = Wire([l3, l2, l1])
left_side = Pos(Z=-0.5) * Shell.extrude(left_wire, (0, 0, 1))
```

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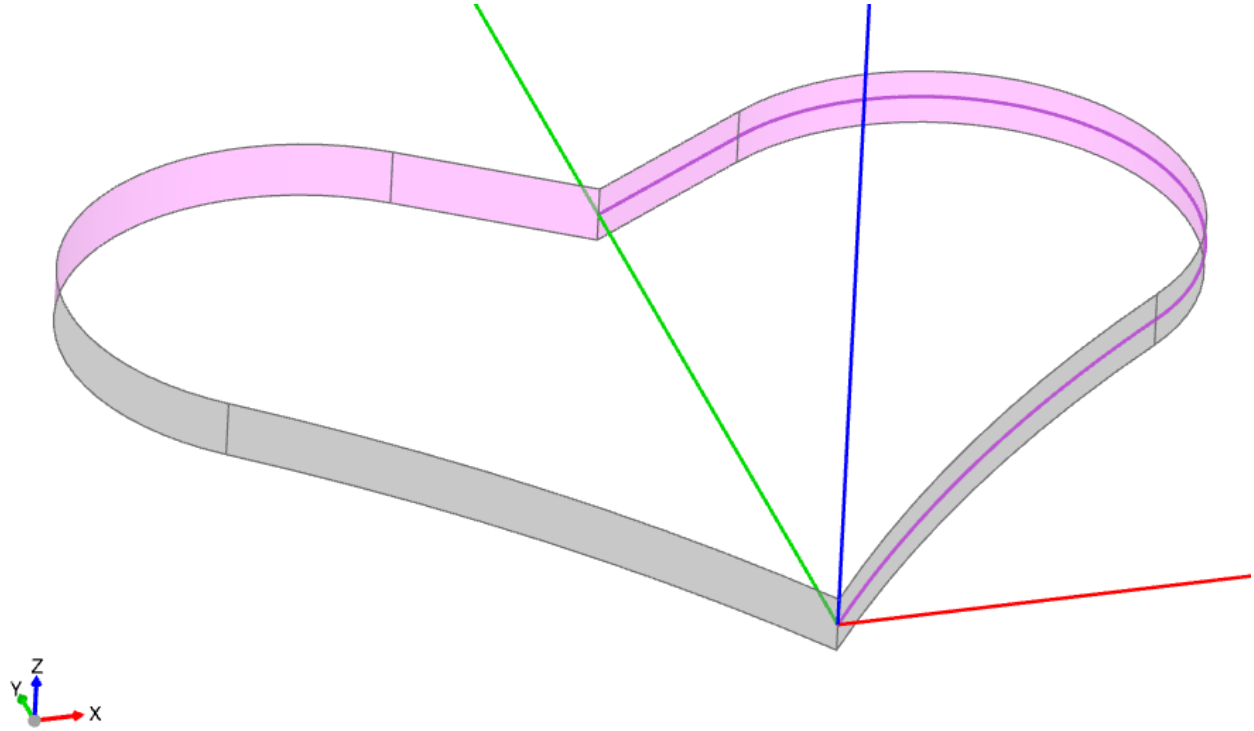
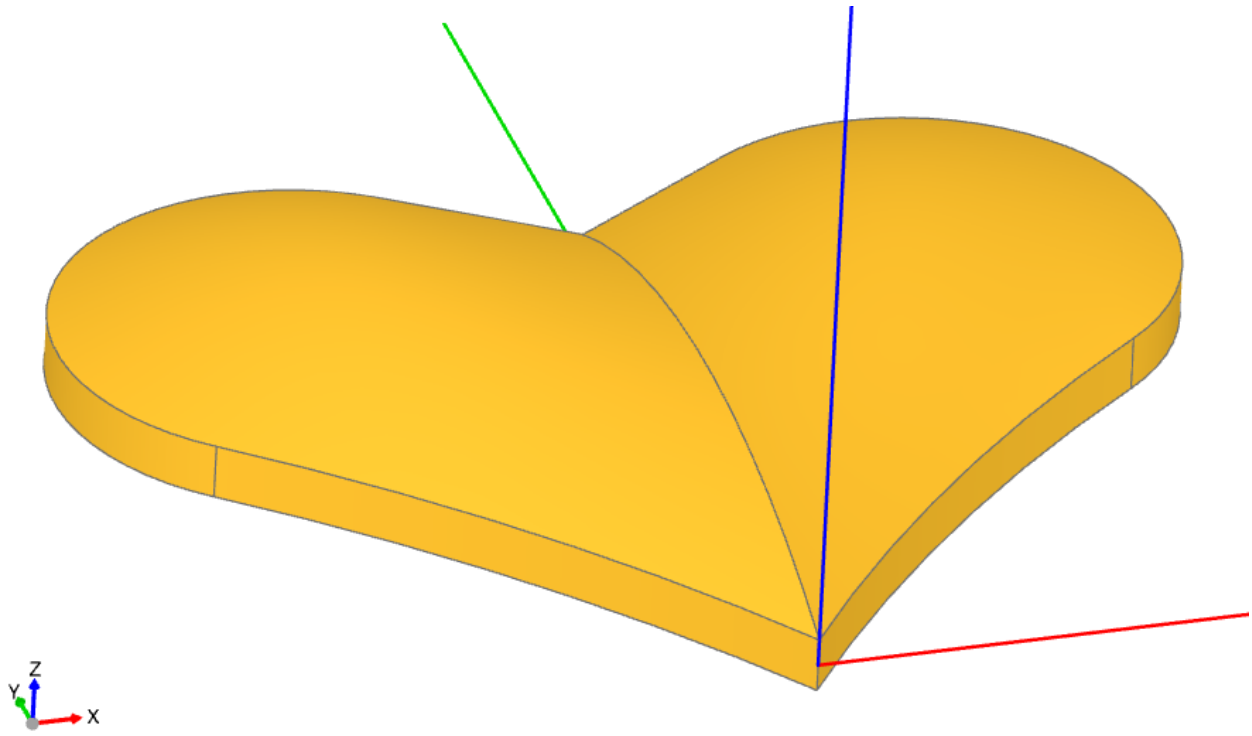
```
right_side = left_side.mirror(Plane.YZ)
```



With the top, bottom, and sides, the complete boundary of the object is defined. We can now put them together, first into a *Shell* and then into a *Solid*:

```
# Put all of the faces together into a Shell/Solid
heart = Solid(
    Shell(
        [
            top_right_surface,
            top_left_surface,
            bottom_right_surface,
            bottom_left_surface,
            left_side,
            right_side,
        ]
    )
)
```



> **Note**
>
> When creating a Solid from a Shell, the Shell must be "water-tight," meaning it should have no holes. For objects with complex Edges, it's best practice to reuse Edges in adjoining Faces whenever possible to avoid slight mismatches that can create openings.

Finally, we'll create the frame around the heart as a simple extrusion of a planar shape defined by the perimeter of the heart and merge all of the components together:

```
# Build a frame around the heart
with BuildPart() as heart_token:
    with BuildSketch() as outline:
        with BuildLine():
            add(l1)
            add(l2)
            add(l3)
            Line(l3 @ 1, l1 @ 0)
        make_face()
        mirror(about=Plane.YZ)
        center = outline.sketch
        offset(amount=2, kind=Kind.INTERSECTION)
        add(center, mode=Mode.SUBTRACT)
    extrude(amount=2, both=True)
    add(heart)

heart_token.part.color = "Red"

show(heart_token)
```

Note that an additional planar line is used to close `l1` and `l3` so a `Face` can be created. The *`offset()`* function defines the outside of the frame as a constant distance from the heart itself.

Summary

In this tutorial, we've explored surface modeling techniques to create a non-planar heart-shaped object using build123d. By utilizing methods from the *`Face`* class, such as *`make_surface()`*, we constructed the perimeter and central point of the surface. We then assembled the complete boundary of the object by creating the top, bottom, and sides, and combined them into a *`Shell`* and eventually a *`Solid`*. Finally, we added a frame around the heart using the *`offset()`* function to maintain a constant distance from the heart.

Next steps

Continue to *Tutorial: Spitfire Wing with Gordon Surface* for an advanced example using *`make_gordon_surface()`* to create a Supermarine Spitfire wing.

Tutorial: Spitfire Wing with Gordon Surface

In this advanced tutorial we construct a Supermarine Spitfire wing as a *`make_gordon_surface()`*—a powerful technique for surfacing from intersecting *profiles* and *guides*. A Gordon surface blends a grid of curves into a smooth, coherent surface as long as the profiles and guides intersect consistently.

> **Note**
>
> Gordon surfaces work best when *each profile intersects each guide exactly once*, producing a well-formed curve network.

Overview

We will:

1. Define overall wing dimensions and elliptic leading/trailing edge guide curves
2. Sample the guides to size the root and tip airfoils (different NACA profiles)
3. Build the Gordon surface from the airfoil *profiles* and wing-edge *guides*
4. Close the root with a planar face and build the final *`Solid`*

Step 1 — Dimensions and guide curves

We model a single wing (half-span), with an elliptic leading and trailing edge. These two edges act as the *guides* for the Gordon surface.

```
from build123d import *
from ocp_vscode import show

wing_span = 36 * FT + 10 * IN
wing_leading = 2.5 * FT
wing_trailing = wing_span / 4 - wing_leading
wing_leading_fraction = wing_leading / (wing_leading + wing_trailing)
wing_tip_section = wing_span / 2 - 1 * IN  # distance from root to last section

# Create leading and trailing edges
leading_edge = EllipticalCenterArc(
```

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```
    (0, 0), wing_span / 2, wing_leading, start_angle=270, end_angle=360
)
trailing_edge = EllipticalCenterArc(
    (0, 0), wing_span / 2, wing_trailing, start_angle=0, end_angle=90
)
```

Step 2 — Root and tip airfoil sizing

We intersect the guides with planes normal to the span to size the airfoil sections. The resulting chord lengths define uniform scales for each airfoil curve.

```
# Calculate the airfoil sizes from the leading/trailing edges
airfoil_sizes = []
for i in [0, 1]:
    tip_axis = Axis(i * (wing_tip_section, 0, 0), (0, 1, 0))
    leading_pnt = leading_edge.intersect(tip_axis)[0]
    trailing_pnt = trailing_edge.intersect(tip_axis)[0]
    airfoil_sizes.append(trailing_pnt.Y - leading_pnt.Y)
```

Step 3 — Build airfoil profiles (root and tip)

We place two different NACA airfoils on `Plane.YZ`—with the airfoil origins shifted so the leading edge fraction is aligned—then scale to the chord lengths from Step 2.

```
# Create the root and tip airfoils - note that they are different NACA profiles
airfoil_root = Plane.YZ * scale(
    Airfoil("2213").translate((-wing_leading_fraction, 0, 0)), airfoil_sizes[0]
)
airfoil_tip = (
    Plane.YZ
    * Pos(Z=wing_tip_section)
    * scale(Airfoil("2205").translate((-wing_leading_fraction, 0, 0)), airfoil_sizes[1])
)
```

Step 4 — Gordon surface construction

A Gordon surface needs *profiles* and *guides*. Here the airfoil edges are the profiles; the elliptic edges are the guides. We also add the wing tip section so the profile grid closes at the tip.

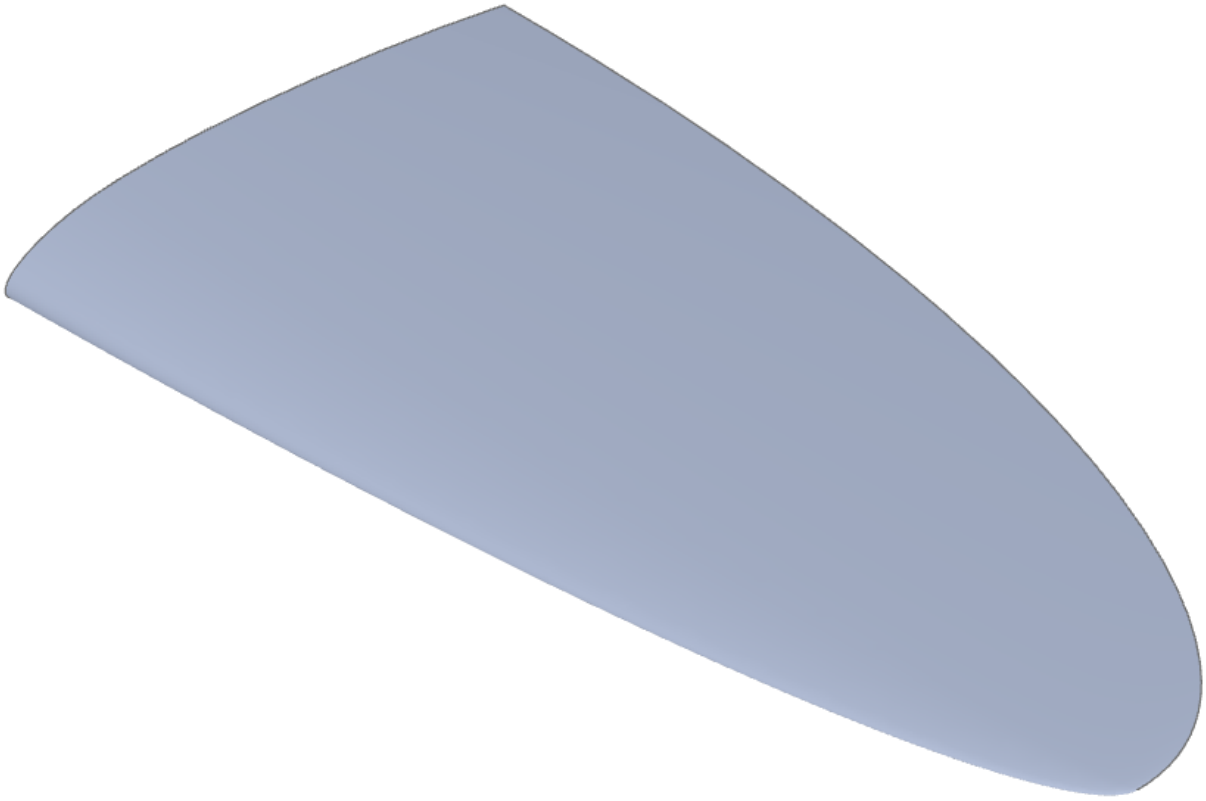
```
# Create the Gordon surface profiles and guides
profiles = airfoil_root.edges() + airfoil_tip.edges()
profiles.append(leading_edge @ 1)  # wing tip
guides = [leading_edge, trailing_edge]
# Create the wing surface as a Gordon Surface
wing_surface = -Face.make_gordon_surface(profiles, guides)
# Create the root of the wing
wing_root = -Face(Wire(wing_surface.edges().filter_by(Edge.is_closed)))
```

Step 5 — Cap the root and create the solid

We extract the closed root edge loop, make a planar cap, and form a solid shell.

```
# Create the wing Solid
wing = Solid(Shell([wing_surface, wing_root]))
wing.color = 0x99A3B9  # Azure Blue

show(wing)
```



Tips for robust Gordon surfaces

- Ensure each profile intersects each guide once and only once
- Keep the curve network coherent (no duplicated or missing intersections)
- When possible, reuse the same *Edge* objects across adjacent faces

Complete listing

For convenience, here is the full script in one block:

```
from build123d import *
from ocp_vscode import show

wing_span = 36 * FT + 10 * IN
```

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```
wing_leading = 2.5 * FT
wing_trailing = wing_span / 4 - wing_leading
wing_leading_fraction = wing_leading / (wing_leading + wing_trailing)
wing_tip_section = wing_span / 2 - 1 * IN  # distance from root to last section

# Create leading and trailing edges
leading_edge = EllipticalCenterArc(
    (0, 0), wing_span / 2, wing_leading, start_angle=270, end_angle=360
)
trailing_edge = EllipticalCenterArc(
    (0, 0), wing_span / 2, wing_trailing, start_angle=0, end_angle=90
)

# [AirfoilSizes]
# Calculate the airfoil sizes from the leading/trailing edges
airfoil_sizes = []
for i in [0, 1]:
    tip_axis = Axis(i * (wing_tip_section, 0, 0), (0, 1, 0))
    leading_pnt = leading_edge.intersect(tip_axis)[0]
    trailing_pnt = trailing_edge.intersect(tip_axis)[0]
    airfoil_sizes.append(trailing_pnt.Y - leading_pnt.Y)

# [Airfoils]
# Create the root and tip airfoils - note that they are different NACA profiles
airfoil_root = Plane.YZ * scale(
    Airfoil("2213").translate((-wing_leading_fraction, 0, 0)), airfoil_sizes[0]
)
airfoil_tip = (
    Plane.YZ
    * Pos(Z=wing_tip_section)
    * scale(Airfoil("2205").translate((-wing_leading_fraction, 0, 0)), airfoil_sizes[1])
)

# [Profiles]
# Create the Gordon surface profiles and guides
profiles = airfoil_root.edges() + airfoil_tip.edges()
profiles.append(leading_edge @ 1)  # wing tip
guides = [leading_edge, trailing_edge]
# Create the wing surface as a Gordon Surface
wing_surface = -Face.make_gordon_surface(profiles, guides)
# Create the root of the wing
wing_root = -Face(Wire(wing_surface.edges().filter_by(Edge.is_closed)))

# [Solid]
# Create the wing Solid
wing = Solid(Shell([wing_surface, wing_root]))
wing.color = 0x99A3B9  # Azure Blue

show(wing)
```

1.9.10 Technical Drawing Tutorial

This example demonstrates how to generate a standard technical drawing of a 3D part using *build123d*. It creates orthographic and isometric views of a Nema 23 stepper motor and exports the result as an SVG file suitable for printing or inspection.

Overview

A technical drawing represents a 3D object in 2D using a series of standardized views. These include:

- **Plan (Top View)** – as seen from directly above (Z-axis down)
- **Front Elevation** – looking at the object head-on (Y-axis forward)
- **Side Elevation (Right Side)** – viewed from the right (X-axis)
- **Isometric Projection** – a 3D perspective view to help visualize depth

Each view is aligned to a position on the page and optionally scaled or annotated.

How It Works

The script uses the *project_to_viewport* method to project the 3D part geometry into 2D. A helper function, *project_to_2d*, sets up the viewport (camera origin and up direction) and places the result onto a virtual drawing sheet.

The steps involved are:

1. Load or construct a 3D part (in this case, a stepper motor).
2. Define a *TechnicalDrawing* border and title block using A4 page size.
3. Generate each of the standard views and apply transformations to place them.
4. Add dimensions using *ExtensionLine* and labels using *Text*.
5. Export the drawing using *ExportSVG*, separating visible and hidden edges by layer and style.

Result

Try It Yourself

You can modify the script to:

- Replace the part with your own *Part* model
- Adjust camera angles and scale
- Add other views (bottom, rear)
- Enhance with more labels and dimensions

Code

```
from datetime import date

from bd_warehouse.open_builds import StepperMotor
from build123d import *
from ocp_vscode import show


def project_to_2d(
```

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```
    part: Part,
    viewport_origin: VectorLike,
    viewport_up: VectorLike,
    page_origin: VectorLike,
    scale_factor: float = 1.0,
) -> tuple[ShapeList[Edge], ShapeList[Edge]]:
    """project_to_2d

    Helper function to generate 2d views translated on the 2d page.

    Args:
        part (Part): 3d object
        viewport_origin (VectorLike): location of viewport
        viewport_up (VectorLike): direction of the viewport Y axis
        page_origin (VectorLike): center of 2d object on page
        scale_factor (float, optional): part scalar. Defaults to 1.0.

    Returns:
        tuple[ShapeList[Edge], ShapeList[Edge]]: visible & hidden edges
    """
    scaled_part = part if scale_factor == 1.0 else scale(part, scale_factor)
    visible, hidden = scaled_part.project_to_viewport(
        viewport_origin, viewport_up, look_at=(0, 0, 0)
    )
    visible = [Pos(*page_origin) * e for e in visible]
    hidden = [Pos(*page_origin) * e for e in hidden]

    return ShapeList(visible), ShapeList(hidden)


# The object that appearing in the drawing
stepper: Part = StepperMotor("Nema23")

# Create a standard technical drawing border on A4 paper
border = TechnicalDrawing(
    designed_by="build123d",
    design_date=date.fromisoformat("2025-05-23"),
    page_size=PageSize.A4,
    title="Nema 23 Stepper",
    sub_title="Units: mm",
    drawing_number="BD-1",
    sheet_number=1,
    drawing_scale=1,
)
page_size = border.bounding_box().size

# Specify the drafting options for extension lines
drafting_options = Draft(font_size=3.5, decimal_precision=1, display_units=False)

# Lists used to store the 2d visible and hidden lines
visible_lines, hidden_lines = [], []
```

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```
# Isometric Projection - A 3D view where the part is rotated to reveal three
# dimensions equally.
iso_v, iso_h = project_to_2d(
    stepper,
    (100, 100, 100),
    (0, 0, 1),
    page_size * 0.3,
    0.75,
)
visible_lines.extend(iso_v)
hidden_lines.extend(iso_h)

# Plan View (Top) - The view from directly above the part (looking down along
# the Z-axis).
vis, _ = project_to_2d(
    stepper,
    (0, 0, 100),
    (0, 1, 0),
    (page_size.X * -0.3, page_size.Y * 0.25),
)
visible_lines.extend(vis)

# Dimension the top of the stepper
top_bbox = Curve(vis).bounding_box()
perimeter = Pos(*top_bbox.center()) * Rectangle(top_bbox.size.X, top_bbox.size.Y)
d1 = ExtensionLine(
    border=perimeter.edges().sort_by(Axis.X)[-1], offset=1 * CM, draft=drafting_options
)
d2 = ExtensionLine(
    border=perimeter.edges().sort_by(Axis.Y)[0], offset=1 * CM, draft=drafting_options
)
# Add a label
l1 = Text("Plan View", 6)
l1.position = vis.sort_by(Axis.Y)[-1].center() + (0, 5 * MM)

# Front Elevation - The primary view, typically looking along the Y-axis,
# showing the height.
vis, _ = project_to_2d(
    stepper,
    (0, -100, 0),
    (0, 0, 1),
    (page_size.X * -0.3, page_size.Y * -0.125),
)
visible_lines.extend(vis)
d3 = ExtensionLine(
    border=vis.sort_by(Axis.Y)[-1], offset=-5 * MM, draft=drafting_options
)
l2 = Text("Front Elevation", 6)
l2.position = vis.group_by(Axis.Y)[0].sort_by(Edge.length)[-1].center() + (0, -5 * MM)

# Side Elevation - Often refers to the Right Side View, looking along the X-axis.
vis, _ = project_to_2d(
```

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```
    stepper,
    (100, 0, 0),
    (0, 0, 1),
    (0, page_size.Y * 0.15),
)
visible_lines.extend(vis)
side_bbox = Curve(vis).bounding_box()
shaft_top_corner = vis.edges().sort_by(Axis.Y)[-1].vertices().sort_by(Axis.X)[-1]
body_bottom_corner = (side_bbox.max.X, side_bbox.min.Y)
d4 = ExtensionLine(
    border=(shaft_top_corner, body_bottom_corner),
    offset=-(side_bbox.max.X - shaft_top_corner.X) - 1 * CM,  # offset to outside view.
    measurement_direction=(0, 1, 0),
    draft=drafting_options,
)
l3 = Text("Side Elevation", 6)
l3.position = vis.group_by(Axis.Y)[0].sort_by(Edge.length)[-1].center() + (0, -5 * MM)


# Initialize the SVG exporter
exporter = ExportSVG(unit=Unit.MM)
# Define visible and hidden line layers
exporter.add_layer("Visible")
exporter.add_layer("Hidden", line_color=(99, 99, 99), line_type=LineType.ISO_DOT)
# Add the objects to the appropriate layer
exporter.add_shape(visible_lines, layer="Visible")
exporter.add_shape(hidden_lines, layer="Hidden")
exporter.add_shape(border, layer="Visible")
exporter.add_shape([d1, d2, d3, d4], layer="Visible")
exporter.add_shape([l1, l2, l3], layer="Visible")
# Write the file
exporter.write(f"assets/stepper_drawing.svg")

show(border, visible_lines, d1, d2, d3, d4, l1, l2, l3)
```

Dependencies

This example depends on the following packages:

- *build123d*
- *bd_warehouse* (for the *StepperMotor* part)
- *ocp_vscode* (for local preview)

1.10 Objects

Objects are Python classes that take parameters as inputs and create 1D, 2D or 3D Shapes. For example, a *Torus* is defined by a major and minor radii. In Builder mode, objects are positioned with `Locations` while in Algebra mode, objects are positioned with the * operator and shown in these examples:

```
with BuildPart() as disk:
    with BuildSketch():
```

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```
        Circle(a)
        with Locations((b, 0.0)):
            Rectangle(c, c, mode=Mode.SUBTRACT)
        with Locations((0, b)):
            Circle(d, mode=Mode.SUBTRACT)
    extrude(amount=c)
```

```
sketch = Circle(a) - Pos(b, 0.0) * Rectangle(c, c) - Pos(0.0, b) * Circle(d)
disk = extrude(sketch, c)
```

The following sections describe the 1D, 2D and 3D objects:

1.10.1 Align

2D/Sketch and 3D/Part objects can be aligned relative to themselves, either centered, or justified right or left of each Axis. The following diagram shows how this alignment works in 2D:

For example:

```
with BuildSketch():
    Circle(1, align=(Align.MIN, Align.MIN))
```

creates a circle who's minimal X and Y values are on the X and Y axis and is located in the top right corner. The `Align` enum has values: `MIN`, `CENTER` and `MAX`.

In 3D the `align` parameter also contains a Z align value but otherwise works in the same way.

Note that the `align` will also accept a single `Align` value which will be used on all axes - as shown here:

```
with BuildSketch():
    Circle(1, align=Align.MIN)
```

1.10.2 Mode

With the Builder API the `mode` parameter controls how objects are combined with lines, sketches, or parts under construction. The `Mode` enum has values:

- `ADD`: fuse this object to the object under construction
- `SUBTRACT`: cut this object from the object under construction
- `INTERSECT`: intersect this object with the object under construction
- `REPLACE`: replace the object under construction with this object
- `PRIVATE`: don't interact with the object under construction at all

The Algebra API doesn't use the `mode` parameter - users combine objects with operators.

1.10.3 1D Objects

The following objects all can be used in BuildLine contexts. Note that 1D objects are not affected by `Locations` in Builder mode.

Airfoil

Airfoil described by 4 digit NACA profile

Bezier

Curve defined by control points and weights

BlendCurve

Curve blending curvature of two curves

BSpline

B-spline from control points and knot data

CenterArc

Arc defined by center, radius, & angles

ConstrainedArcs

Arc(s) constrained by other geometric objects

ConstrainedLines

Line(s) constrained by other geometric objects

DoubleTangentArc

Arc defined by point/tangent pair & other curve

EllipticalCenterArc

Elliptical arc defined by center, radii & angles

EllipticalStartArc

Elliptical arc defined by start, tangent, radii & angles

ParabolicCenterArc

Parabolic arc defined by vertex, focal length & angles

HyperbolicCenterArc

Hyperbolic arc defined by center, radii & angles

FilletPolyline

Polyline with filleted corners defined by pts and radius

Helix

Helix defined pitch, radius and height

IntersectingLine

Intersecting line defined by start, direction & other line

JernArc

Arc define by start point, tangent, radius and angle

Line

Line defined by end points

PolarLine

Line defined by start, angle and length

Polyline

Multiple line segments defined by points

RadiusArc

Arc defined by two points and a radius

SagittaArc

Arc defined by two points and a sagitta

Spline

Curve define by points

TangentArc

Arc defined by two points and a tangent

ThreePointArc

Arc defined by three points

ArcArcTangentLine

Line tangent defined by two arcs

ArcArcTangentArc

Arc tangent defined by two arcs

PointArcTangentLine

Line tangent defined by a point and arc

PointArcTangentArc

Arc tangent defined by a point, direction, and arc

Reference

class BaseLineObject(*curve: ~build123d.topology.one_d.Wire*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

BaseLineObject specialized for Wire.

Parameters

- **curve** (Wire) – wire to create
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class Airfoil(*airfoil_code: str*, *n_points: int = 50*, *finite_te: bool = False*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Create an airfoil described by a 4-digit (or fractional) NACA airfoil (e.g. '2412' or '2213.323').

The NACA four-digit wing sections define the airfoil_code by: - First digit describing maximum camber as percentage of the chord. - Second digit describing the distance of maximum camber from the airfoil leading edge in tenths of the chord. - Last two digits describing maximum thickness of the airfoil as percent of the chord.

Parameters

- **airfoil_code** – str The NACA 4-digit (or fractional) airfoil code (e.g. '2213.323').
- **n_points** – int Number of points per upper/lower surface.
- **finite_te** – bool If True, enforces a finite trailing edge (default False).
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

property camber_line: Edge

Camber line of the airfoil as an Edge.

camber_pos: float

Chordwise position of max camber (0–1)

code: str

NACA code string (e.g. "2412")

finite_te: bool

If True, trailing edge is finite

max_camber: float

Maximum camber as fraction of chord

static parse_naca4(*value: str | float*) → tuple[float, float, float]

Parse NACA 4-digit (or fractional) airfoil code into parameters.

thickness: float

Maximum thickness as fraction of chord

class Bezier(**cntl_pnts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], weights: list[float] | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Bezier Curve

Create a non-rational bezier curve defined by a sequence of points and include optional weights to create a rational bezier curve. The number of weights must match the number of control points.

Parameters

- **cntl_pnts** (*sequence[VectorLike]*) – points defining the curve
- **weights** (*list[float], optional*) – control point weights. Defaults to None
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class BlendCurve(*curve0: ~build123d.topology.one_d.Edge, curve1: ~build123d.topology.one_d.Edge, continuity: ~build123d.build_enums.ContinuityLevel = ContinuityLevel.C2, end_points: tuple[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]] | None = None, tangent_scalars: tuple[float, float] | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: BlendCurve

Create a smooth Bézier-based transition curve between two existing edges.

The blend is constructed as a cubic (C1) or quintic (C2) Bézier curve whose control points are determined from the position, first derivative, and (for C2) second derivative of the input curves at the chosen endpoints. Optional scalar multipliers can be applied to the endpoint tangents to control the "tension" of the blend.

Parameters

- **curve0** (Edge) – First curve to blend from.
- **curve1** (Edge) – Second curve to blend to.
- **continuity** (*ContinuityLevel, optional*) – Desired geometric continuity at the join: - ContinuityLevel.C0: position match only (straight line) - ContinuityLevel.C1: match position and tangent direction (cubic Bézier) - ContinuityLevel.C2: match position, tangent, and curvature (quintic Bézier) Defaults to ContinuityLevel.C2.

- **end_points** (*tuple[VectorLike, VectorLike] | None, optional*) – Pair of points specifying the connection points on *curve0* and *curve1*. Each must coincide (within TOLERANCE) with the start or end of the respective curve. If None, the closest pair of endpoints is chosen. Defaults to None.
- **tangent_scalars** (*tuple[float, float] | None, optional*) – Scalar multipliers applied to the first derivatives at the start of *curve0* and the end of *curve1* before computing control points. Useful for adjusting the pull/tension of the blend without altering the base curves. Defaults to (1.0, 1.0).
- **mode** (*Mode, optional*) – Boolean operation mode when used in a BuildLine context. Defaults to Mode.ADD.

Raises

- **ValueError** – *tangent_scalars* must be a pair of float values.
- **ValueError** – If specified *end_points* are not coincident with the start or end of their respective curves.

Example

```
>>> blend = BlendCurve(curve_a, curve_b, ContinuityLevel.C1, tangent_scalars=(1.2,␣
↪0.8))
>>> show(blend)
```

class BSpline(*control_points: ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], knots: ~collections.abc.Iterable[float], degree: int, weights: ~collections.abc.Iterable[float] | None = None, periodic: bool = False, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: BSpline

An exact B-spline edge defined directly from control points and knot data.

BSpline creates an exact B-spline from control points, a knot sequence, and optional weights. Control points define the control polygon that pulls the curve, but the curve does not generally pass through them. Knots define the parameter-space structure of the spline: they determine where polynomial spans begin and end and how smoothly those spans join. Repeated knot values indicate knot multiplicity. For a spline of degree p, a knot with multiplicity m has continuity C^(p-m) at that location, so increasing multiplicity reduces smoothness. Repeating the first and last knots degree + 1 times creates a clamped spline that starts and ends at the first and last control points. Optional weights create a rational B-spline, allowing some control points to pull more strongly than others and enabling exact representation of conic sections.`

Unlike `Spline`, which creates an interpolated curve through a set of points using `GeomAPI_Interpolate`, `BSpline` preserves the supplied spline definition by building the underlying OCCT `Geom_BSplineCurve` from its poles, knot vector, optional weights, degree, and periodic flag.

Parameters

- **control_points** (*Iterable[VectorLike]*) – Control points (poles) defining the spline shape. These are not generally points on the curve.
- **knots** (*Iterable[float]*) – Knot sequence for the spline. Repeated knot values are allowed and are converted internally into unique knot values plus multiplicities as required by OCCT.
- **degree** (*int*) – Polynomial degree of the spline.
- **weights** (*Iterable[float] | None, optional*) – Optional per-control-point weights for rational B-splines. If omitted, the spline is non-rational.

- **periodic** (*bool, optional*) – Whether to create a periodic spline. Defaults to `False`.
- **mode** (*Mode, optional*) – Builder combination mode. Defaults to `Mode.ADD`.

class CenterArc(*center: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], radius: float, start_angle: float, arc_size: float | ~build123d.topology.shape_core.Shape | ~build123d.geometry.Axis | ~build123d.geometry.Location | ~build123d.geometry.Plane | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Center Arc

Create a circular arc defined by a center point and radius.

Parameters

- **center** (*VectorLike*) – center point of arc
- **radius** (*float*) – arc radius
- **start_angle** (*float*) – arc starting angle from x-axis
- **arc_size** (*float | Shape | Axis | Location | Plane | VectorLike*) – angular size of arc or an arc limit.

  When a limit object is provided instead of a numeric angular size, CenterArc constructs the valid arc(s) from the given start point, trims them at their first intersection with the limit, and returns the one requiring the shortest travel from the start. Therefore, one can only generate arcs < 180° using a limit. If neither valid arc intersects the limit, a ValueError is raised.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class ConstrainedArcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], tangency_two: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], *, radius: float, sagitta: Sagitta = Sagitta.SHORT, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda arcs: ..., mode: Mode = Mode.ADD*)

class ConstrainedArcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], tangency_two: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], *, center_on: Axis | Edge, sagitta: Sagitta = Sagitta.SHORT, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda arcs: ..., mode: Mode = Mode.ADD*)

class ConstrainedArcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], tangency_two: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], tangency_three: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], *, sagitta: Sagitta = Sagitta.SHORT, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda arcs: ..., mode: Mode = Mode.ADD*)

class ConstrainedArcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], *, center: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda arcs: ..., mode: Mode = Mode.ADD*)

class ConstrainedArcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], *, radius: float, center_on: Edge, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda arcs: ..., mode: Mode = Mode.ADD*)

Line Object: Arc(s) constrained by other geometric objects.

The result is always a Curve containing one or more Edges. If you need to access Edge-specific properties or methods (such as `arc_center`), extract the edge or edges first:

```
result = ConstrainedArcs(...)
arc = result.edge()           # extract the Edge
center = arc.arc_center       # now Edge methods are available
```

Note that in Builder mode the `selector` parameter must be provided or all results will be combined into the BuildLine context. In Algebra mode the selector can be applied as a parameter or in the normal way to the ConstrainedArcs object. The content of the selector is the same in both cases.

Examples

An arc built from three edge constraints.

Algebra:

```
l4 = PolarLine((0, 0), 4, 60)
l5 = PolarLine((0, 0), 4, 40)
a3 = CenterArc((0, 0), 4, 0, 90)
ex_a3 = (
    ConstrainedArcs(l4, l5, a3, sagitta=Sagitta.BOTH).edges().sort_by(Edge.
↪length)[0]
)
```

Builder:

```
with BuildLine() as arc_ex3:
    l4 = PolarLine((0, 0), 4, 60)
    l5 = PolarLine((0, 0), 4, 40)
    a3 = CenterArc((0, 0), 4, 0, 90)
    ex_a3 = ConstrainedArcs(
        l4,
        l5,
        a3,
        sagitta=Sagitta.BOTH,
        selector=lambda arcs: arcs.sort_by(Edge.length)[0],
    )
```

class ConstrainedLines(*tangency_one: tuple[Edge, Tangency] | Axis | Edge, tangency_two: tuple[Edge, Tangency] | Axis | Edge, *, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda lines: ..., mode: Mode = Mode.ADD*)

class ConstrainedLines(*tangency_one: tuple[Edge, Tangency] | Edge, tangency_two: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float], *, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda lines: ..., mode: Mode = Mode.ADD*)

class ConstrainedLines(*tangency_one: tuple[Edge, Tangency] | Edge, tangency_two: Axis, *, angle: float | None = None, direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None, selector: Callable[[ShapeList[Edge]], Edge | ShapeList[Edge]] = lambda lines: ..., mode: Mode = Mode.ADD*)

Line Object: Lines(s) constrained by other geometric objects.

The result is always a Curve containing one or more Edges. If you need to access Edge-specific properties or methods (such as `length`), extract the edge or edges first:

```
result = ConstrainedLines(...)
lines = result.edges()      # extract the Edges
length = lines[1].length    # now Edge methods are available
```

Note that in Builder mode the `selector` parameter must be provided or all results will be combined into the BuildLine context. In Algebra mode the selector can be applied as a parameter or in the normal way to the ConstrainedArcs object. The content of the selector is the same in both cases.

class DoubleTangentArc(*pnt: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], tangent: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], other: ~build123d.topology.composite.Curve | ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire, keep: ~build123d.build_enums.Keep = <Keep.TOP>, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Double Tangent Arc

Create a circular arc defined by a point/tangent pair and another line find a tangent to.

The arc specified with TOP or BOTTOM depends on the geometry and isn't predictable.

Contains a solver.

Parameters

- **pnt** (*VectorLike*) – start point
- **tangent** (*VectorLike*) – tangent at start point
- **other** (Curve | Edge | Wire) – line object to tangent
- **keep** (Keep, *optional*) – specify which arc if more than one, TOP or BOTTOM. Defaults to Keep.TOP
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Raises

RunTimeError – no double tangent arcs found

class EllipticalCenterArc(*center: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], x_radius: float, y_radius: float, start_angle: float = 0.0, end_angle: float | None = None, *, arc_size: float | ~build123d.topology.shape_core.Shape | ~build123d.geometry.Axis | ~build123d.geometry.Location | ~build123d.geometry.Plane | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] = 90.0, rotation: float = 0.0, angular_direction: ~build123d.build_enums.AngularDirection | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Elliptical Center Arc

Create an elliptical arc defined by a center point, x- and y- radii.

Parameters

- **center** (*VectorLike*) – ellipse center
- **x_radius** (*float*) – x radius of the ellipse (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the ellipse (along the y-axis of plane)
- **start_angle** (*float, optional*) – arc start angle from x-axis. Defaults to 0.0
- **end_angle** (*float | None*) – arc end angle from x-axis. Defaults to None

- **arc_size** (*float | Shape | Axis | Location | Plane | VectorLike*) – angular size of arc (negative to change direction) or an arc limit.

  When a limit object is provided instead of a numeric angular size, EllipticalCenterArc constructs the valid arc(s) from the given start point, trims them at their first intersection with the limit, and returns the one requiring the shortest travel from the start. Therefore, one can only generate arcs < 180° using a limit. If neither valid arc intersects the limit, a ValueError is raised.

- **rotation** (*float, optional*) – angle to rotate arc. Defaults to 0.0
- **angular_direction** (*AngularDirection | None*) – arc direction. Defaults to None.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class EllipticalStartArc(*start_pnt: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], start_tangent: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], x_radius: float, y_radius: float, arc_size: float, *, start_angle: float | None = None, major_axis_dir: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: EllipticalStartArc

Create a circular arc defined by a start point/tangent pair, radius and arc size.

Parameters

- **start_pnt** (*VectorLike*) – start point
- **start_tangent** (*VectorLike*) – tangent at start point
- **x_radius** (*float*) – x radius of the ellipse (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the ellipse (along the y-axis of plane)
- **arc_size** (*float*) – angular size of arc (negative to change direction)
- **start_angle** (*float*) – angular position of the start point
- **major_axis_dir** (*VectorLike*) – direction of ellipse x-axis
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

Note

One of start_angle or major_axis_dir must be provided.

class ParabolicCenterArc(*vertex: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], focal_length: float, start_angle: float = 0.0, end_angle: float | None = None, *, arc_size: float | ~build123d.topology.shape_core.Shape | ~build123d.geometry.Axis | ~build123d.geometry.Location | ~build123d.geometry.Plane | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] = 90.0, rotation: float = 0.0, angular_direction: ~build123d.build_enums.AngularDirection | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Parabolic Center Arc

Create a parabolic arc defined by a vertex point and focal length (distance from focus to vertex).

Parameters

- **vertex** (*VectorLike*) – parabola vertex
- **focal_length** (*float*) – focal length the parabola (distance from the vertex to focus along the x-axis of plane)
- **start_angle** (*float, optional*) – arc start angle. Defaults to 0.0
- **end_angle** (*float | None, optional*) – arc end angle. Defaults to None
- **arc_size** (*float | Shape | Axis | Location | Plane | VectorLike*) – angular size of arc (negative to change direction) or an arc limit.

  When a limit object is provided instead of a numeric angular size, ParabolicCenterArc constructs candidate arcs from the given start point, trims them at their first intersection with the limit, and returns the one requiring the shortest travel from the start. If neither valid arc intersects the limit, a ValueError is raised.
- **rotation** (*float, optional*) – angle to rotate arc. Defaults to 0.0
- **angular_direction** (*AngularDirection | None, optional*) – arc direction. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class HyperbolicCenterArc(*center: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], x_radius: float, y_radius: float, start_angle: float = 0.0, end_angle: float | None = None, *, arc_size: float | ~build123d.topology.shape_core.Shape | ~build123d.geometry.Axis | ~build123d.geometry.Location | ~build123d.geometry.Plane | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] = 90.0, rotation: float = 0.0, angular_direction: ~build123d.build_enums.AngularDirection | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Hyperbolic Center Arc

Create a hyperbolic arc defined by a center point and focal length (distance from focus to vertex).

Parameters

- **center** (*VectorLike*) – hyperbola center
- **x_radius** (*float*) – x radius of the ellipse (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the ellipse (along the y-axis of plane)
- **start_angle** (*float, optional*) – arc start angle from x-axis. Defaults to 0.0
- **end_angle** (*float | None, optional*) – arc end angle from x-axis. Defaults to None
- **arc_size** (*float | Shape | Axis | Location | Plane | VectorLike*) – angular size of arc (negative to change direction) or an arc limit.

  When a limit object is provided instead of a numeric angular size, HyperbolicCenterArc constructs candidate arcs from the given start point, trims them at their first intersection with the limit, and returns the one requiring the shortest travel from the start. If neither valid arc intersects the limit, a ValueError is raised.
- **rotation** (*float, optional*) – angle to rotate arc. Defaults to 0.0
- **angular_direction** (*AngularDirection | None, optional*) – arc direction. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class FilletPolyline(*pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], radius: float | ~collections.abc.Iterable[float], close: bool = False, mode: ~build123d.build_enums.Mode = <Mode.ADD>)

Line Object: Fillet Polyline Create a sequence of straight lines defined by successive points that are filleted to a given radius.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of two or more points
- **radius** (*float | Iterable[float]*) – radius to fillet at each vertex or a single value for all vertices. A radius of 0 will create a sharp corner (vertex without fillet).
- **close** (*bool, optional*) – close end points with extra Edge and corner fillets. Defaults to False
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Raises

- **ValueError** – Two or more points not provided
- **ValueError** – radius must be non-negative

class Helix(*pitch: float, height: float, radius: float, center: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] = (0, 0, 0), direction: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] = (0, 0, 1), cone_angle: float = 0, lefthand: bool = False, mode: ~build123d.build_enums.Mode = <Mode.ADD>)

Line Object: Helix

Create a helix defined by pitch, height, and radius. The helix may have a taper defined by cone_angle.

If cone_angle is not 0, radius is the initial helix radius at center. cone_angle > 0 increases the final radius. cone_angle < 0 decreases the final radius.

Parameters

- **pitch** (*float*) – distance between loops
- **height** (*float*) – helix height
- **radius** (*float*) – helix radius
- **center** (*VectorLike, optional*) – center point. Defaults to (0, 0, 0)
- **direction** (*VectorLike, optional*) – direction of central axis. Defaults to (0, 0, 1)
- **cone_angle** (*float, optional*) – conical angle from direction. Defaults to 0
- **lefthand** (*bool, optional*) – left handed helix. Defaults to False
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class IntersectingLine(*start: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], direction: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], other: ~build123d.topology.composite.Curve | ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire, mode: ~build123d.build_enums.Mode = <Mode.ADD>)

Intersecting Line Object: Line

Create a straight line defined by a point/direction pair and another line to intersect.

Parameters

- **start** (*VectorLike*) – start point
- **direction** (*VectorLike*) – direction to make line
- **other** (Edge) – line object to intersect
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class JernArc(*start: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], tangent: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], radius: float, arc_size: float | ~build123d.topology.shape_core.Shape | ~build123d.geometry.Axis | ~build123d.geometry.Location | ~build123d.geometry.Plane | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Jern Arc

Create a circular arc defined by a start point/tangent pair, radius and arc size or arc limit.

Parameters

- **start** (*VectorLike*) – start point
- **tangent** (*VectorLike*) – tangent at start point
- **radius** (*float*) – arc radius
- **arc_size** (*float | Shape | Axis | Location | Plane | VectorLike*) – angular size of arc (negative to change direction) or an arc limit.

  When a limit object is provided instead of a numeric angular size, JernArc constructs the valid tangent arc(s) from the given start point and tangent, trims them at their first intersection with the limit, and returns the one requiring the shortest travel from the start. If neither valid arc intersects the limit, a ValueError is raised.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Variables

- **start** (Vector) – start point
- **end_of_arc** (Vector) – end point of arc
- **center_point** (Vector) – center of arc

class Line(**pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Line

Create a straight line defined by two points.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of two points
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – Two point not provided

class PolarLine(*start: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], length: float | ~build123d.topology.shape_core.Shape | ~build123d.geometry.Axis | ~build123d.geometry.Location | ~build123d.geometry.Plane | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], angle: float | None = None, direction: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None, length_mode: ~build123d.build_enums.LengthMode = <LengthMode.DIAGONAL>, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Polar Line

Create a straight line defined by a start point, length, and angle. The length can specify the DIAGONAL, HORIZONTAL, or VERTICAL component of the triangle defined by the angle.

Alternatively, the length parameter can contain a limit to the length of the line in the form of another object. If the PolarLine doesn't contact the limit an error will be generated.

Example

p = PolarLine(start=(2, 0), length=Axis.Y, angle=135)

Parameters

- **start** (*VectorLike*) – start point
- **length** (*float | Shape | Axis | Location | Plane | VectorLike*) – line length (float) or limit limit
- **angle** (*float, optional*) – angle from the local x-axis
- **direction** (*VectorLike, optional*) – vector direction to determine angle
- **length_mode** (*LengthMode, optional*) – how length defines the line. Defaults to LengthMode.DIAGONAL
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

Raises

- **ValueError** – Either angle or direction must be provided
- **ValueError** – Polar line doesn't intersect length limit

class Polyline(**pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], close: bool = False, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Polyline

Create a sequence of straight lines defined by successive points.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of two or more points
- **close** (*bool, optional*) – close by generating an extra Edge. Defaults to False
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – Two or more points not provided

class RadiusArc(*start_point: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], end_point: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], radius: float, short_sagitta: bool = True, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Radius Arc

Create a circular arc defined by two points and a radius.

Parameters

- **start_point** (*VectorLike*) – start point
- **end_point** (*VectorLike*) – end point
- **radius** (*float*) – arc radius
- **short_sagitta** (*bool*) – If True selects the short sagitta (height of arc from chord), else the long sagitta crossing the center. Defaults to True
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – Insufficient radius to connect end points

class SagittaArc(*start_point: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], end_point: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], sagitta: float, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Sagitta Arc

Create a circular arc defined by two points and the sagitta (height of the arc from chord).

Parameters

- **start_point** (*VectorLike*) – start point
- **end_point** (*VectorLike*) – end point
- **sagitta** (*float*) – arc height from chord between points
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class Spline(**pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], tangents: ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]] | None = None, tangent_scalars: ~collections.abc.Iterable[float] | None = None, periodic: bool = False, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Line Object: Spline

Create a spline defined by a sequence of points, optionally constrained by tangents. Tangents and tangent scalars must have length of 2 for only the end points or a length of the number of points.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of two or more points
- **tangents** (*Iterable[VectorLike], optional*) – tangent directions. Defaults to None
- **tangent_scalars** (*Iterable[float], optional*) – tangent scales. Defaults to None
- **periodic** (*bool, optional*) – make the spline periodic (closed). Defaults to False
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class TangentArc(*pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], tangent: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], tangent_from_first: bool = True, mode: ~build123d.build_enums.Mode = <Mode.ADD>)

Line Object: Tangent Arc

Create a circular arc defined by two points and a tangent.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of two points
- **tangent** (*VectorLike*) – tangent to constrain arc
- **tangent_from_first** (*bool, optional*) – apply tangent to first point. Applying tangent to end point will flip the orientation of the arc. Defaults to True
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – Two points are required

class ThreePointArc(*pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], mode: ~build123d.build_enums.Mode = <Mode.ADD>)

Line Object: Three Point Arc

Create a circular arc defined by three points.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of three points
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – Three points must be provided

class ArcArcTangentLine(**kwargs)

Line Object: Arc Arc Tangent Line

Create a straight line tangent to two arcs.

Parameters

- **start_arc** (Curve | Edge | Wire) – starting arc, must be GeomType.CIRCLE
- **end_arc** (Curve | Edge | Wire) – ending arc, must be GeomType.CIRCLE
- **side** (*Side*) – side of arcs to place tangent arc center, LEFT or RIGHT. Defaults to Side.LEFT
- **keep** (Keep) – which tangent arc to keep, INSIDE or OUTSIDE. Defaults to Keep.INSIDE
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class ArcArcTangentArc(**kwargs)

Line Object: Arc Arc Tangent Arc

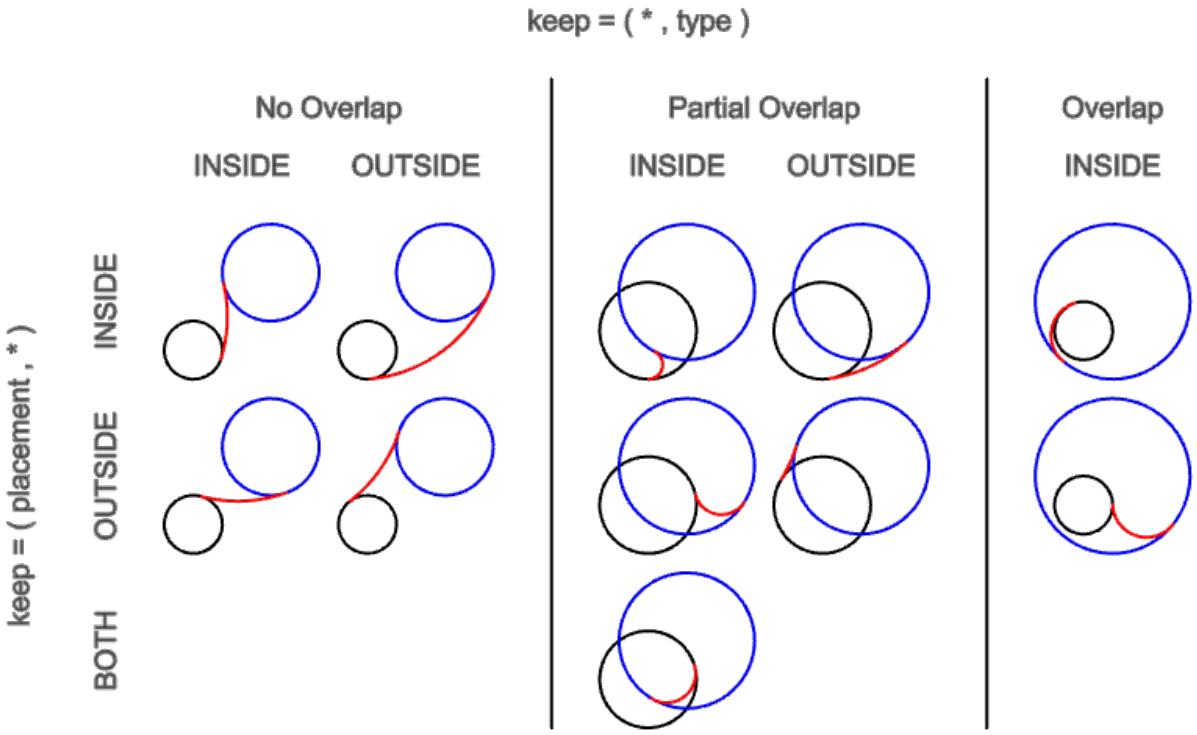
Create an arc tangent to two arcs and a radius.

keep specifies tangent arc position with a Keep pair: (placement, type)

- placement: start_arc is tangent INSIDE or OUTSIDE the tangent arc. BOTH is a special case for overlapping arcs with type INSIDE
- type: tangent arc is INSIDE or OUTSIDE start_arc and end_arc

Parameters

- **start_arc** (Curve | Edge | Wire) – starting arc, must be GeomType.CIRCLE
- **end_arc** (Curve | Edge | Wire) – ending arc, must be GeomType.CIRCLE
- **radius** (*float*) – radius of tangent arc
- **side** (*Side*) – side of arcs to place tangent arc center, LEFT or RIGHT. Defaults to Side.LEFT
- **keep** (Keep | *tuple*[Keep, Keep]) – which tangent arc to keep, INSIDE or OUTSIDE. Defaults to (Keep.INSIDE, Keep.INSIDE)
- **short_sagitta** (*bool*) – If True selects the short sagitta (height of arc from chord), else the long sagitta crossing the center. Defaults to True
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD



class PointArcTangentLine(***kwargs*)

Line Object: Point Arc Tangent Line

Create a straight, tangent line from a point to a circular arc.

Parameters

- **point** (*VectorLike*) – intersection point for tangent
- **arc** (Curve | Edge | Wire) – circular arc to tangent, must be GeomType.CIRCLE

- **side** (*Side, optional*) – side of arcs to place tangent arc center, LEFT or RIGHT. Defaults to Side.LEFT
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class PointArcTangentArc(***kwargs*)

Line Object: Point Arc Tangent Arc

Create an arc defined by a point/tangent pair and another line which the other end is tangent to.

Parameters

- **point** (*VectorLike*) – starting point of tangent arc
- **direction** (*VectorLike*) – direction at starting point of tangent arc
- **arc** (*Union[Curve, Edge, Wire]*) – ending arc, must be GeomType.CIRCLE
- **side** (*Side, optional*) – select which arc to keep Defaults to Side.LEFT
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

Raises

- **ValueError** – Arc must have GeomType.CIRCLE
- **ValueError** – Point is already tangent to arc
- **RuntimeError** – No tangent arc found

1.10.4 2D Objects

Arrow

Arrow with head and path for shaft

ArrowHead

Arrow head with multiple types

Circle

Circle defined by radius

DimensionLine

Dimension line

Ellipse

Ellipse defined by major and minor radius

ExtensionLine

Extension lines for distance or angles

Polygon

Polygon defined by points

Rectangle

Rectangle defined by width and height

RectangleRounded

Rectangle with rounded corners defined by width, height, and radius

RegularPolygon

RegularPolygon defined by radius and number of sides

SlotArc

SlotArc defined by arc and height

SlotCenterPoint

SlotCenterPoint defined by two points and a height

SlotCenterToCenter

SlotCenterToCenter defined by center separation and height

SlotOverall

SlotOverall defined by end-to-end length and height

TechnicalDrawing

A technical drawing with descriptions

Text

Text defined by string and font parameters

Trapezoid

Trapezoid defined by width, height and interior angles

Triangle

Triangle defined by one side & two other sides or interior angles

Reference

class BaseSketchObject(*obj: ~build123d.topology.composite.Compound | ~build123d.topology.two_d.Face*, *rotation: float = 0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Base class for all BuildSketch objects

Parameters

- **face** (*Face*) – face to create
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (*Align | tuple[Align, Align], optional*) – align MIN, CENTER, or MAX of object. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class Arrow(*arrow_size: float*, *shaft_path: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire*, *shaft_width: float*, *head_at_start: bool = True*, *head_type: ~build123d.build_enums.HeadType = <HeadType.CURVED>*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Arrow with shaft

Parameters

- **arrow_size** (*float*) – arrow head tip to tail length
- **shaft_path** (*Edge | Wire*) – line describing the shaft shape
- **shaft_width** (*float*) – line width of shaft
- **head_at_start** (*bool, optional*) – Defaults to True.

- **head_type** (*HeadType, optional*) – arrow head shape. Defaults to HeadType.CURVED.
- **mode** (Mode, *optional*) – _description_. Defaults to Mode.ADD.

class ArrowHead(*size: float*, *head_type: ~build123d.build_enums.HeadType = <HeadType.CURVED>*, *rotation: float = 0*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: ArrowHead

Parameters

- **size** (*float*) – tip to tail length
- **head_type** (*HeadType, optional*) – arrow head shape. Defaults to HeadType.CURVED.
- **rotation** (*float, optional*) – rotation in degrees. Defaults to 0.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

class Circle(*radius: float*, *arc_size: float = 360.0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = (<Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Circle

Create a circle defined by radius.

Parameters

- **radius** (*float*) – circle radius
- **arc_size** (*float, optional*) – angular size of sector. Defaults to 360.
- **align** (Align | *tuple[*Align, Align*], optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class DimensionLine(*path: ~build123d.topology.one_d.Wire | ~build123d.topology.one_d.Edge | list[~build123d.geometry.Vector | ~build123d.topology.zero_d.Vertex | tuple[float, float, float]], draft: ~drafting.Draft, sketch: ~build123d.topology.composite.Sketch | None = None, label: str | None = None, arrows: tuple[bool, bool] = (True, True), tolerance: float | tuple[float, float] | None = None, label_angle: bool = False, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: DimensionLine

Create a dimension line typically for internal measurements. Typically used for (but not restricted to) inside dimensions, a dimension line often as arrows on either side of a dimension or label.

There are three options depending on the size of the text and length of the dimension line: Type 1) The label and arrows fit within the length of the path Type 2) The text fit within the path and the arrows go outside Type 3) Neither the text nor the arrows fit within the path

Parameters

- **path** (*PathDescriptor*) – a very general type of input used to describe the path the dimension line will follow.
- **draft** (*Draft*) – instance of Draft dataclass
- **sketch** (Sketch) – the Sketch being created to check for possible overlaps. In builder mode the active Sketch will be used if None is provided.

- **label** (*str, optional*) – a text string which will replace the length (or arc length) that would otherwise be extracted from the provided path. Providing a label is useful when illustrating a parameterized input where the name of an argument is desired not an actual measurement. Defaults to None.
- **arrows** (*tuple[bool, bool], optional*) – a pair of boolean values controlling the placement of the start and end arrows. Defaults to (True, True).
- **tolerance** (*float | tuple[float, float], optional*) – an optional tolerance value to add to the extracted length value. If a single tolerance value is provided it is shown as ± the provided value while a pair of values are shown as separate + and - values. Defaults to None.
- **label_angle** (*bool, optional*) – a flag indicating that instead of an extracted length value, the size of the circular arc extracted from the path should be displayed in degrees.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

Raises

- **ValueError** – Only 2 points allowed for dimension lines
- **ValueError** – No output - no arrows selected

dimension

length of the dimension

class Ellipse(*x_radius: float*, *y_radius: float*, *rotation: float = 0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = (<Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Ellipse

Create an ellipse defined by x- and y- radii.

Parameters

- **x_radius** (*float*) – x radius of the ellipse (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the ellipse (along the y-axis of plane)
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (Align | *tuple[*Align, Align*], optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class ExtensionLine(*border: ~build123d.topology.one_d.Wire | ~build123d.topology.one_d.Edge | list[~build123d.geometry.Vector | ~build123d.topology.zero_d.Vertex | tuple[float, float, float]], offset: float, draft: ~drafting.Draft, sketch: ~build123d.topology.composite.Sketch | None = None, label: str | None = None, arrows: tuple[bool, bool] = (True, True), tolerance: float | tuple[float, float] | None = None, label_angle: bool = False, measurement_direction: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Extension Line

Create a dimension line with two lines extending outward from the part to dimension. Typically used for (but not restricted to) outside dimensions, with a pair of lines extending from the edge of a part to a dimension line.

Parameters

- **border** (*PathDescriptor*) – a very general type of input defining the object to be dimensioned. Typically this value would be extracted from the part but is not restricted to this use.
- **offset** (*float*) – a distance to displace the dimension line from the edge of the object
- **draft** (*Draft*) – instance of Draft dataclass
- **label** (*str, optional*) – a text string which will replace the length (or arc length) that would otherwise be extracted from the provided path. Providing a label is useful when illustrating a parameterized input where the name of an argument is desired not an actual measurement. Defaults to None.
- **arrows** (*tuple[bool, bool], optional*) – a pair of boolean values controlling the placement of the start and end arrows. Defaults to (True, True).
- **tolerance** (*float | tuple[float, float], optional*) – an optional tolerance value to add to the extracted length value. If a single tolerance value is provided it is shown as ± the provided value while a pair of values are shown as separate + and - values. Defaults to None.
- **label_angle** (*bool, optional*) – a flag indicating that instead of an extracted length value, the size of the circular arc extracted from the path should be displayed in degrees. Defaults to False.
- **measurement_direction** (*VectorLike, optional*) – Vector line which to project the dimension against. Offset start point is the position of the start of border. Defaults to None.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

dimension

length of the dimension

class **Polygon**(**pts: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], rotation: float = 0, align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = (<Align.NONE>, <Align.NONE>), mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Polygon

Create a polygon defined by given sequence of points.

Note: the order of the points defines the resulting normal of the Face in Algebra mode, where counter-clockwise order creates an upward normal while clockwise order a downward normal. In Builder mode, the Face is added with an upward normal.

Parameters

- **pts** (*VectorLike | Iterable[VectorLike]*) – sequence of points defining the vertices of the polygon
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (Align | *tuple[*Align, Align*], optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.NONE, Align.NONE)
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class **Rectangle**(*width: float*, *height: float*, *rotation: float = 0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align*, *~build123d.build_enums.Align] | None = (<Align.CENTER>*, *<Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Rectangle

Create a rectangle defined by width and height.

Parameters

- **width** (*float*) – rectangle width
- **height** (*float*) – rectangle height
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (Align | tuple[Align, Align], optional) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (Mode, optional) – combination mode. Defaults to Mode.ADD

class RectangleRounded(*width: float*, *height: float*, *radius: float*, *rotation: float = 0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = (<Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Rectangle Rounded

Create a rectangle defined by width and height with filleted corners.

Parameters

- **width** (*float*) – rectangle width
- **height** (*float*) – rectangle height
- **radius** (*float*) – fillet radius
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (Align | tuple[Align, Align], optional) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (Mode, optional) – combination mode. Defaults to Mode.ADD

class RegularPolygon(*radius: float*, *side_count: int*, *major_radius: bool = True*, *rotation: float = 0*, *align: tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Regular Polygon

Create a regular polygon defined by radius and side count. Use major_radius to define whether the polygon circumscribes (along the vertices) or inscribes (along the sides) the radius circle.

Parameters

- **radius** (*float*) – construction radius
- **side_count** (*int*) – number of sides
- **major_radius** (*bool*) – If True the radius is the major radius (circumscribed circle), else the radius is the minor radius (inscribed circle). Defaults to True
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (Align | tuple[Align, Align], optional) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (Mode, optional) – combination mode. Defaults to Mode.ADD

apothem: float

radius of the inscribed circle or minor radius

radius: float

radius of the circumscribed circle or major radius

class SlotArc(*arc: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire*, *height: float*, *rotation: float = 0*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Slot Arc

Create a slot defined by a line and height. May be an arc, stright line, spline, etc.

Parameters

- **arc** (Edge | Wire) – center line of slot
- **height** (*float*) – diameter of end arcs
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class SlotCenterPoint(*center: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], point: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], height: float, rotation: float = 0, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Slot Center Point

Create a slot defined by the center of the slot and the center of one end arc. The slot will be symmetric about the center point.

Parameters

- **center** (*VectorLike*) – center point
- **point** (*VectorLike*) – center of arc point
- **height** (*float*) – diameter of end arcs
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class SlotCenterToCenter(*center_separation: float*, *height: float*, *rotation: float = 0*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Slot Center To Center

Create a slot defined by the distance between the centers of the two end arcs.

Parameters

- **center_separation** (*float*) – distance between arc centers
- **height** (*float*) – diameter of end arcs
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class SlotOverall(*width: float*, *height: float*, *rotation: float = 0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = (<Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Slot Overall

Create a slot defined by the overall width and height.

Parameters

- **width** (*float*) – overall width of slot
- **height** (*float*) – diameter of end arcs
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (Align | tuple[Align, Align], optional) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class TechnicalDrawing(*designed_by: str = 'build123d'*, *design_date: ~datetime.date | None = None*, *page_size: ~build123d.build_enums.PageSize = <PageSize.A4>*, *title: str = 'Title'*, *sub_title: str = 'Sub Title'*, *drawing_number: str = 'B3D-1'*, *sheet_number: int | None = None*, *drawing_scale: float = 1.0*, *nominal_text_size: float = 10.0*, *line_width: float = 0.5*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: TechnicalDrawing

The border of a technical drawing with external frame and text box.

Parameters

- **designed_by** (*str, optional*) – Defaults to "build123d".
- **design_date** (*date, optional*) – Defaults to date.today().
- **page_size** (*PageSize, optional*) – Defaults to PageSize.A4.
- **title** (*str, optional*) – drawing title. Defaults to "Title".
- **sub_title** (*str, optional*) – drawing sub title. Defaults to "Sub Title".
- **drawing_number** (*str, optional*) – Defaults to "B3D-1".
- **sheet_number** (*int, optional*) – Defaults to None.
- **drawing_scale** (*float, optional*) – displays as 1:value. Defaults to 1.0.
- **nominal_text_size** (*float, optional*) – size of title text. Defaults to 10.0.
- **line_width** (*float, optional*) – Defaults to 0.5.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

margin = 5

page_sizes = {<PageSize.A0>: (1189, 841), <PageSize.A10>: (37, 26), <PageSize.A1>: (841, 594), <PageSize.A2>: (594, 420), <PageSize.A3>: (420, 297), <PageSize.A4>: (297, 210), <PageSize.A5>: (210, 148.5), <PageSize.A6>: (148.5, 105), <PageSize.A7>: (105, 74), <PageSize.A8>: (74, 52), <PageSize.A9>: (52, 37), <PageSize.LEDGER>: (431.79999999999995, 279.4), <PageSize.LEGAL>: (355.59999999999997, 215.89999999999998), <PageSize.LETTER>: (279.4, 215.89999999999998)}

class Text(*txt: str*, *font_size: float*, *font: str = 'Arial'*, *font_path: ~os.PathLike[str] | str | None = None*, *font_style: ~build123d.build_enums.FontStyle = <FontStyle.REGULAR>*, *text_align: tuple[~build123d.build_enums.TextAlign, ~build123d.build_enums.TextAlign] = (<TextAlign.CENTER>, <TextAlign.CENTER>)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = None*, *path: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | None = None*, *position_on_path: float = 0.0*, *single_line_width: float | None = None*, *rotation: float = 0.0*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Text

Create text defined by text string and font size.

Fonts installed to the system can be specified by name and FontStyle. Fonts with subfamilies not in FontStyle should be specified with the subfamily name, e.g. "Arial Black". Alternatively, a specific font file can be specified with font_path.

Use *available_fonts()* to list available font names for *font* and FontStyles. Note: on Windows, fonts must be installed with "Install for all users" to be found by name.

Not all fonts have every FontStyle available, however ITALIC and BOLDITALIC will still italicize the font if the respective font file is not available.

text_align specifies alignment of text inside the bounding box, while align the aligns the bounding box itself.

Optionally, the Text can be positioned on a non-linear edge or wire with a path and position_on_path.

Parameters

- **txt** (*str*) – text to render
- **font_size** (*float*) – size of the font in model units
- **font** (*str, optional*) – font name. Defaults to "Arial"
- **font_path** (*PathLike | str, optional*) – system path to font file. Defaults to None
- **font_style** (*Font_Style, optional*) – font style, REGULAR, BOLD, BOLDITALIC, or ITALIC. Defaults to Font_Style.REGULAR
- **text_align** (*tuple[TextAlign, TextAlign], optional*) – horizontal text align LEFT, CENTER, or RIGHT. Vertical text align BOTTOM, CENTER, TOP, or TOPFIRST-LINE. Defaults to (TextAlign.CENTER, TextAlign.CENTER)
- **align** (Align | *tuple[*Align, Align*], optional*) – align MIN, CENTER, or MAX of object. Defaults to None
- **path** (Edge | Wire, *optional*) – path for text to follow. Defaults to None
- **position_on_path** (*float, optional*) – the relative location on path to position the text, values must be between 0.0 and 1.0. Defaults to 0.0
- **single_line_width** (*float, optional*) – width of outlined single line font. Defaults to 4% of font_size
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD

class Trapezoid(*width: float*, *height: float*, *left_side_angle: float*, *right_side_angle: float | None = None*, *rotation: float = 0*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = (<Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Trapezoid

Create a trapezoid defined by major width, height, and interior angle(s).

Parameters

- **width** (*float*) – trapezoid major width
- **height** (*float*) – trapezoid height
- **left_side_angle** (*float*) – bottom left interior angle
- **right_side_angle** (*float, optional*) – bottom right interior angle. If not provided, the trapezoid will be symmetric. Defaults to None
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (*Align | tuple[Align, Align], optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER)
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – Give angles result in an invalid trapezoid

class **Triangle**(*, *a: float | None = None*, *b: float | None = None*, *c: float | None = None*, *A: float | None = None*, *B: float | None = None*, *C: float | None = None*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = None*, *rotation: float = 0*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Sketch Object: Triangle

Create a triangle defined by one side length and any of two other side lengths or interior angles. The interior angles are opposite the side with the same designation (i.e. side 'a' is opposite angle 'A'). Side 'a' is the bottom side, followed by 'b' on the right, going counter-clockwise.

Parameters

- **a** (*float, optional*) – side 'a' length. Defaults to None
- **b** (*float, optional*) – side 'b' length. Defaults to None
- **c** (*float, optional*) – side 'c' length. Defaults to None
- **A** (*float, optional*) – interior angle 'A'. Defaults to None
- **B** (*float, optional*) – interior angle 'B'. Defaults to None
- **C** (*float, optional*) – interior angle 'C'. Defaults to None
- **rotation** (*float, optional*) – angle to rotate object. Defaults to 0
- **align** (*Align | tuple[Align, Align], optional*) – align MIN, CENTER, or MAX of object. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

Raises

ValueError – One length and two other values were not provided

A

interior angle 'A' in degrees

B

interior angle 'B' in degrees

C
: interior angle 'C' in degrees

a
: length of side 'a'

b
: length of side 'b'

c
: length of side 'c'

edge_a
: edge 'a'

edge_b
: edge 'b'

edge_c
: edge 'c'

vertex_A
: vertex 'A'

vertex_B
: vertex 'B'

vertex_C
: vertex 'C'

1.10.5 3D Objects

Box

Box defined by length, width, height

Cone

Cone defined by radii and height

ConvexPolyhedron

Convex Polyhedron defined by points

CounterBoreHole

Counter bore hole defined by radii and depths

CounterSinkHole

Counter sink hole defined by radii and depth and angle

Cylinder

Cylinder defined by radius and height

Hole

Hole defined by radius and depth

Sphere

Sphere defined by radius and arc angles

Torus

Torus defined major and minor radii

Wedge

Wedge defined by lengths along multiple Axes

Reference

class BasePartObject(*part: ~build123d.topology.composite.Part | ~build123d.topology.three_d.Solid*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Base class for all BuildPart objects & operations

Parameters

- **solid** (*Solid*) – object to create
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)
- **align** (*Align | tuple[Align, Align, Align] | None, optional*) – align MIN, CENTER, or MAX of object. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD

class Box(*length: float*, *width: float*, *height: float*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: Box

Create a box defined by length, width, and height.

Parameters

- **length** (*float*) – box length
- **width** (*float*) – box width
- **height** (*float*) – box height
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)
- **align** (*Align | tuple[Align, Align, Align] | None, optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER, Align.CENTER)
- **mode** (*Mode, optional*) – combine mode. Defaults to Mode.ADD

class Cone(*bottom_radius: float*, *top_radius: float*, *height: float*, *arc_size: float = 360*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: Cone

Create a cone defined by bottom radius, top radius, and height.

Parameters

- **bottom_radius** (*float*) – bottom radius

- **top_radius** (*float*) – top radius, may be zero
- **height** (*float*) – cone height
- **arc_size** (*float, optional*) – angular size of cone. Defaults to 360
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)
- **align** (*Align | tuple[Align, Align, Align] | None, optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER, Align.CENTER)
- **mode** (*Mode, optional*) – combine mode. Defaults to Mode.ADD

class ConvexPolyhedron(*points: ~collections.abc.Iterable[~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float]], rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0), align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = <Align.NONE>, mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: ConvexPolyhedron

Create a convex solid from the convex hull of the provided points.

Parameters

- **points** (*Iterable[VectorLike]*) – vertices of the polyhedron
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)
- **align** (*Align | tuple[Align, Align, Align] | None, optional*) – align MIN, CENTER, or MAX of object. Defaults to Align.NONE
- **mode** (*Mode, optional*) – combine mode. Defaults to Mode.ADD

class CounterBoreHole(*radius: float*, *counter_bore_radius: float*, *counter_bore_depth: float*, *depth: float | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.SUBTRACT>*)

Part Operation: Counter Bore Hole

Create a counter bore hole defined by radius, counter bore radius, counter bore and depth.

Parameters

- **radius** (*float*) – hole radius
- **counter_bore_radius** (*float*) – counter bore radius
- **counter_bore_depth** (*float*) – counter bore depth
- **depth** (*float, optional*) – hole depth, through part if None. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.SUBTRACT

class CounterSinkHole(*radius: float*, *counter_sink_radius: float*, *depth: float | None = None*, *counter_sink_angle: float = 82*, *mode: ~build123d.build_enums.Mode = <Mode.SUBTRACT>*)

Part Operation: Counter Sink Hole

Create a countersink hole defined by radius, countersink radius, countersink angle, and depth.

Parameters

- **radius** (*float*) – hole radius
- **counter_sink_radius** (*float*) – countersink radius

- **depth** (*float, optional*) – hole depth, through part if None. Defaults to None
- **counter_sink_angle** (*float, optional*) – cone angle. Defaults to 82
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.SUBTRACT

class Cylinder(*radius: float*, *height: float*, *arc_size: float = 360*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: Cylinder

Create a cylinder defined by radius and height.

Parameters

- **radius** (*float*) – cylinder radius
- **height** (*float*) – cylinder height
- **arc_size** (*float, optional*) – angular size of cone. Defaults to 360.
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)
- **align** (*Align | tuple[Align, Align, Align] | None, optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER, Align.CENTER)
- **mode** (*Mode, optional*) – combine mode. Defaults to Mode.ADD

class Hole(*radius: float*, *depth: float | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.SUBTRACT>*)

Part Operation: Hole

Create a hole defined by radius and depth.

Parameters

- **radius** (*float*) – hole radius
- **depth** (*float, optional*) – hole depth, through part if None. Defaults to None
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.SUBTRACT

class Sphere(*radius: float*, *arc_size1: float = -90*, *arc_size2: float = 90*, *arc_size3: float = 360*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: Sphere

Create a sphere defined by a radius.

Parameters

- **radius** (*float*) – sphere radius
- **arc_size1** (*float, optional*) – angular size of bottom hemisphere. Defaults to -90.
- **arc_size2** (*float, optional*) – angular size of top hemisphere. Defaults to 90.
- **arc_size3** (*float, optional*) – angular revolution about pole. Defaults to 360.
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)

- **align** (Align | tuple[Align, Align, Align] | None, optional) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER, Align.CENTER)
- **mode** (Mode, optional) – combine mode. Defaults to Mode.ADD

class Torus(*major_radius: float*, *minor_radius: float*, *minor_start_angle: float = 0*, *minor_end_angle: float = 360*, *major_angle: float = 360*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: Torus

Create a torus defined by major and minor radii.

Parameters

- **major_radius** (*float*) – major torus radius
- **minor_radius** (*float*) – minor torus radius
- **minor_start_angle** (*float, optional*) – angle to start minor arc. Defaults to 0
- **minor_end_angle** (*float, optional*) – angle to end minor arc. Defaults to 360
- **major_angle** (*float, optional*) – angle to revolve minor arc. Defaults to 360
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)
- **align** (Align | tuple[Align, Align, Align] | None, optional) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER, Align.CENTER)
- **mode** (Mode, optional) – combine mode. Defaults to Mode.ADD

class Wedge(*xsize: float*, *ysize: float*, *zsize: float*, *xmin: float*, *zmin: float*, *xmax: float*, *zmax: float*, *rotation: ~build123d.geometry.Rotation | tuple[float, float, float] = (0, 0, 0)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>, <Align.CENTER>)*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

Part Object: Wedge

Create a wedge with a near face defined by xsize and z size, a far face defined by xmin to xmax and zmin to zmax, and a depth of ysize.

Parameters

- **xsize** (*float*) – length of near face along x-axis
- **ysize** (*float*) – length of part along y-axis
- **zsize** (*float*) – length of near face z-axis
- **xmin** (*float*) – minimum position far face along x-axis
- **zmin** (*float*) – minimum position far face along z-axis
- **xmax** (*float*) – maximum position far face along x-axis
- **zmax** (*float*) – maximum position far face along z-axis
- **rotation** (*RotationLike, optional*) – angles to rotate about axes. Defaults to (0, 0, 0)

- **align** (*Align | tuple[Align, Align, Align] | None, optional*) – align MIN, CENTER, or MAX of object. Defaults to (Align.CENTER, Align.CENTER, Align.CENTER)
- **mode** (*Mode, optional*) – combine mode. Defaults to Mode.ADD

1.10.6 Text

Create Text Object

Create text object or add to `BuildSketch` using *Text*:


```
text = "The quick brown fox jumped over the lazy dog."
Text(text, 10)
```

Specify font and style. Fonts have up to 4 font styles: `REGULAR`, `BOLD`, `ITALIC`, `BOLDITALIC`. All fonts can use `ITALIC` even if only `REGULAR` is defined.

```
Text(text, 10, "Arial", font_style=FontStyle.BOLD)
```

Find available fonts on system and available styles:

```
from pprint import pprint
pprint(available_fonts())
```

```
[
 ...
 Font(name='Arial', styles=('REGULAR', 'BOLD', 'BOLDITALIC', 'ITALIC')),
 Font(name='Arial Black', styles=('REGULAR',)),
 Font(name='Arial Narrow', styles=('REGULAR', 'BOLD', 'BOLDITALIC', 'ITALIC')),
 Font(name='Arial Rounded MT Bold', styles=('REGULAR',)),
 ...
]
```

Font faces like `"Arial Black"` or `"Arial Narrow"` must be specified by name rather than `FontStyle`:

```
Text(text, 10, "Arial Black")
```

Specify a font file directly by filename:

```
Text(text, 10, font_path="DejaVuSans.ttf")
```

Fonts added via `font_path` persist in the font list:

```
Text(text, 10, font_path="SourceSans3-VariableFont_wght.ttf")
pprint([f.name for f in available_fonts() if "Source Sans" in f.name])
Text(text, 10, "Source Sans 3 Medium")
```

```
['Source Sans 3',
 'Source Sans 3 Black',
 'Source Sans 3 ExtraBold',
 'Source Sans 3 ExtraLight',
 ...]
```

Add a font file to `FontManager` if a font is reused in the script or contains multiple font faces:

```
new_font_faces = FontManager().register_font("Roboto-VariableFont_wdth,wght.ttf")
pprint(new_font_faces)
Text(text, 10, "Roboto")
Text(text, 10, "Roboto Black")
```

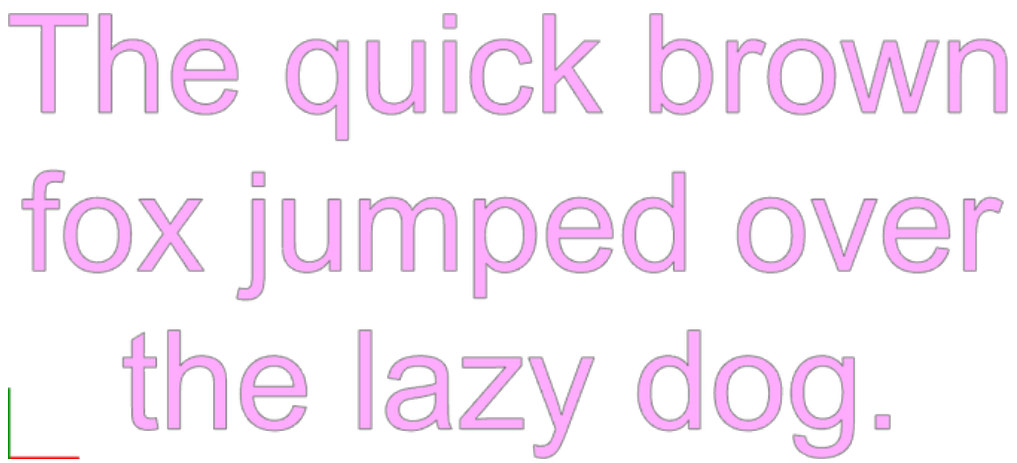
```
['Roboto Thin',
 'Roboto ExtraLight',
 'Roboto Light',
 'Roboto',
  ...]
```

Placement

Multiline text has two methods of alignment. `text_align` aligns the text relative to its `Location`:


```
Text(text, 10, text_align=(TextAlign.LEFT, TextAlign.TOPFIRSTLINE))
```

`align` aligns the object bounding box relative to its `Location` *after* text alignment:




```
text = "The quick brown\nfox jumped over\nthe lazy dog."
Text(text, 10, align=(Align.MIN, Align.MIN))
```

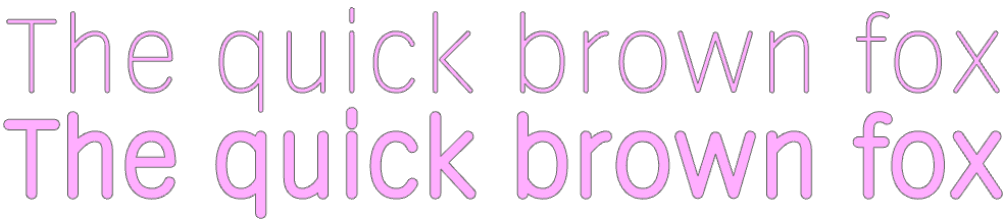
Place text along an `Edge` or `Wire` with `path` and `position_on_path`:


```
text = "The quick brown fox"
path = RadiusArc((-50, 0), (50, 0), 100)
Text(
    text,
    10,
    path=path,
    position_on_path=.5,
    text_align=(TextAlign.CENTER, TextAlign.BOTTOM)
)
```

Single Line Fonts

`"singleline"` is a special font referencing `Relief SingleLine CAD`. Glyphs are represented as single lines rather than filled faces.

`Text` creates an outlined face by default. The outline width is controlled by `single_line_width`. This operation is slow with many glyphs.




```
Text(text, 10, "singleline")
Text(text, 10, "singleline", single_line_width=1)
```

Use `Compound.make_text()` to create *unoutlined* single-line text. Useful for routing, engraving, or drawing label paths.


```
Compound.make_text(text, 10, "singleline")
```

Common Issues

Missing Glyphs or Invalid Geometry

Modern variable-width fonts often contain glyphs with overlapping stroke outlines, which produce invalid geometry. `ocp_vscode` ignores invalid faces.

```
Text("The", 10, "Source Sans 3 Black")
```

FileNotFoundError

Ensure relative `font_path` specifications are relative to the *current working directory*.

1.10.7 Custom Objects

All of the objects presented above were created using one of three base object classes: *BaseLineObject* , *BaseSketchObject* , and *BasePartObject* . Users can use these base object classes to easily create custom objects that have all the functionality of the core objects.

Here is an example of a custom sketch object specially created as part of the design of this playing card storage box (`see the playing_cards.py example`):

```
class Club(BaseSketchObject):
    def __init__(
        self,
        height: float,
        rotation: float = 0,
        align: tuple[Align, Align] = (Align.CENTER, Align.CENTER),
        mode: Mode = Mode.ADD,
    ):
        with BuildSketch() as club:
            with BuildLine():
                l0 = Line((0, -188), (76, -188))
                b0 = Bezier(l0 @ 1, (61, -185), (33, -173), (17, -81))
                b1 = Bezier(b0 @ 1, (49, -128), (146, -145), (167, -67))
                b2 = Bezier(b1 @ 1, (187, 9), (94, 52), (32, 18))
                b3 = Bezier(b2 @ 1, (92, 57), (113, 188), (0, 188))
                mirror(about=Plane.YZ)
            make_face()
            scale(by=height / club.sketch.bounding_box().size.Y)
        super().__init__(obj=club.sketch, rotation=rotation, align=align, mode=mode)
```

Here the new custom object class is called `Club` and it's a sub-class of *BaseSketchObject* . The `__init__` method contains all of the parameters used to instantiate the custom object, specially a `height`, `rotation`, `align`, and `mode` - your objects may contain a sub or super set of these parameters but should always contain a `mode` parameter such that it can be combined with a builder's object.

Next is the creation of the object itself, in this case a sketch of the club suit.

The final line calls the `__init__` method of the super class - i.e. *BaseSketchObject* with its parameters.

That's it, now the `Club` object can be used anywhere a *Circle* would be used - with either the Algebra or Builder API.

1.11 Operations

Operations are functions that take objects as inputs and transform them into new objects. For example, a 2D Sketch can be extruded to create a 3D Part. All operations are Python functions which can be applied using both the Algebra and Builder APIs. It's important to note that objects created by operations are not affected by `Locations`, meaning their position is determined solely by the input objects used in the operation.

Here are a couple ways to use *extrude()*, in Builder and Algebra mode:

```
with BuildPart() as cylinder:
    with BuildSketch():
        Circle(radius)
    extrude(amount=height)
```

```
cylinder = extrude(Circle(radius), amount=height)
```

The following table summarizes all of the available operations. Operations marked as 1D are applicable to BuildLine and Algebra Curve, 2D to BuildSketch and Algebra Sketch, 3D to BuildPart and Algebra Part.

Operation	Description	0D	1D	2D	3D	Example
add()	Add object to builder		✓	✓	✓	*16*
bounding_box()	Add bounding box as Shape		✓	✓	✓	
chamfer()	Bevel Vertex or Edge			✓	✓	*9*
draft()	Add a draft taper to a part				✓	*Cast Bearing Unit*
extrude()	Draw 2D Shape into 3D				✓	*3*
fillet()	Radius Vertex or Edge			✓	✓	*9*
full_round()	Round-off Face along given Edge			✓		*24-SPO-06 Buffer Stand*
loft()	Create 3D Shape from sections				✓	*24*
make_brake_formed()	Create sheet metal parts				✓	
make_face()	Create a Face from Edges			✓		*4*
make_hull()	Create Convex Hull from Edges			✓		
mirror()	Mirror about Plane		✓	✓	✓	*15*
offset()	Inset or outset Shape		✓	✓	✓	*25*
project()	Project points, lines or Faces	✓	✓	✓		
project_workplane()	Create workplane for projection					
revolve()	Swing 2D Shape about Axis				✓	*23*
scale()	Change size of Shape		✓	✓	✓	
section()	Generate 2D slices from 3D Shape				✓	
split()	Divide object by Plane		✓	✓	✓	*27*
sweep()	Extrude 1/2D section(s) along path			✓	✓	*14*
thicken()	Expand 2D section(s)				✓	
trace()	Convert lines to faces			✓		

The following table summarizes all of the selectors that can be used within the scope of a Builder. Note that they will extract objects from the builder that is currently within scope without it being explicitly referenced.

Selector	Description	Builder Line	Sketch	Part
edge()	Select edge from current builder	✓	✓	✓
edges()	Select edges from current builder	✓	✓	✓
face()	Select face from current builder		✓	✓
faces()	Select faces from current builder		✓	✓
solid()	Select solid from current builder			✓
solids()	Select solids from current builder			✓
vertex()	Select vertex from current builder	✓	✓	✓
vertices()	Select vertices from current builder	✓	✓	✓
wire()	Select wire from current builder	✓	✓	✓
wires()	Select wires from current builder	✓	✓	✓

1.11.1 Reference

add(*objects: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid | ~build123d.topology.composite.Compound | ~build123d.build_common.Builder | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid | ~build123d.topology.composite.Compound | ~build123d.build_common.Builder], rotation: float | ~build123d.geometry.Rotation | tuple[float, float, float] | None = None, clean: bool = True, mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Compound

Generic Object: Add Object to Part or Sketch

Add an object to a builder.

BuildPart:
Edges and Wires are added to pending_edges. Compounds of Face are added to pending_faces. Solids or Compounds of Solid are combined into the part.

BuildSketch:
Edges and Wires are added to pending_edges. Compounds of Face are added to sketch.

BuildLine:
Edges and Wires are added to line.

Parameters

- **objects** (*Edge | Wire | Face | Solid | Compound or Iterable of*) – objects to add
- **rotation** (*float | RotationLike, optional*) – rotation angle for sketch, rotation about each axis for part. Defaults to None.
- **clean** – Remove extraneous internal structure. Defaults to True.

bounding_box(*objects: ~build123d.topology.shape_core.Shape | ~collections.abc.Iterable[~build123d.topology.shape_core.Shape] | None = None, mode: ~build123d.build_enums.Mode = <Mode.PRIVATE>*) → Sketch | Part

Generic Operation: Add Bounding Box

Applies to: BuildSketch and BuildPart

Add the 2D or 3D bounding boxes of the object sequence

Parameters

- **objects** (*Shape or Iterable of*) – objects to create bbox for
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

chamfer(*objects: Edge | Vertex | Iterable[Edge | Vertex]*, *length: float*, *length2: float | None = None*, *angle: float | None = None*, *reference: Edge | Face | None = None*) → Sketch | Part

Generic Operation: chamfer

Applies to 2 and 3 dimensional objects.

Chamfer the given sequence of edges or vertices.

Parameters

- **objects** (*Edge | Vertex or Iterable of*) – edges or vertices to chamfer
- **length** (*float*) – chamfer size
- **length2** (*float, optional*) – asymmetric chamfer size. Defaults to None.
- **angle** (*float, optional*) – chamfer angle in degrees. Defaults to None.
- **reference** (*Edge | Face*) – identifies the side where length is measured. Edge(s) must be part of the face. Vertex/Vertices must be part of edge

Raises

- **ValueError** – no objects provided
- **ValueError** – objects must be Edges
- **ValueError** – objects must be Vertices
- **ValueError** – Only one of length2 or angle should be provided
- **ValueError** – reference can only be used in conjunction with length2 or angle

draft(*faces: Face | Iterable[Face]*, *neutral_plane: Plane*, *angle: float*) → Part

Part Operation: draft

Apply a draft angle to the given faces of the part

Parameters

- **faces** – Faces to which the draft should be applied.
- **neutral_plane** – Plane defining the neutral direction and position.
- **angle** – Draft angle in degrees.

extrude(*to_extrude: ~build123d.topology.two_d.Face | ~build123d.topology.composite.Sketch | None = None*, *amount: float | None = None*, *dir: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None*, *until: ~build123d.build_enums.Until | None = None*, *target: ~build123d.topology.three_d.Solid | ~build123d.topology.composite.Compound | None = None*, *both: bool = False*, *taper: float = 0.0*, *clean: bool = True*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Part

Part Operation: extrude

Extrude a sketch or face by an amount or until another object.

Parameters

- **to_extrude** (*Union[Face, Sketch], optional*) – object to extrude. Defaults to None.
- **amount** (*float, optional*) – distance to extrude, sign controls direction. Defaults to None.

- **dir** (*VectorLike, optional*) – direction. Defaults to None.
- **until** (Until, *optional*) – extrude limit. Defaults to None.
- **target** (Shape, *optional*) – extrude until target. Defaults to None.
- **both** (*bool, optional*) – extrude in both directions. Defaults to False.
- **taper** (*float, optional*) – taper angle. Defaults to 0.0.
- **clean** (*bool, optional*) – Remove extraneous internal structure. Defaults to True.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

Raises

- **ValueError** – No object to extrude
- **ValueError** – No target object

Returns
extruded object

Return type
Part

fillet(*objects: Edge | Vertex | Iterable[Edge | Vertex]*, *radius: float*) → Sketch | Part | Curve

Generic Operation: fillet

Applies to 2 and 3 dimensional objects.

Fillet the given sequence of edges or vertices. Note that vertices on either end of an open line will be automatically skipped.

Parameters

- **objects** (Edge | Vertex *or Iterable of*) – edges or vertices to fillet
- **radius** (*float*) – fillet size - must be less than 1/2 local width

Raises

- **ValueError** – no objects provided
- **ValueError** – objects must be Edges
- **ValueError** – objects must be Vertices
- **ValueError** – nothing to fillet

full_round(*edge: ~build123d.topology.one_d.Edge*, *invert: bool = False*, *voronoi_point_count: int = 100*, *mode: ~build123d.build_enums.Mode = <Mode.REPLACE>*) → tuple[Sketch, Vector, float]

Sketch Operation: full_round

Given an edge from a Face/Sketch, modify the face by replacing the given edge with the arc of the Voronoi largest empty circle that will fit within the Face. This "rounds off" the end of the object.

Parameters

- **edge** (Edge) – target Edge to remove
- **invert** (*bool, optional*) – make the arc concave instead of convex. Defaults to False.
- **voronoi_point_count** (*int, optional*) – number of points along each edge used to create the voronoi vertices as potential locations for the center of the largest empty circle. Defaults to 100.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.REPLACE.

Raises
ValueError – Invalid geometry

Returns
the modified shape

Return type
Sketch

loft(*sections: ~build123d.topology.two_d.Face | ~build123d.topology.composite.Sketch | ~collections.abc.Iterable[~build123d.topology.zero_d.Vertex | ~build123d.topology.two_d.Face | ~build123d.topology.composite.Sketch] | None = None*, *ruled: bool = False*, *clean: bool = True*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Part

Part Operation: loft

Loft the pending sketches/faces, across all workplanes, into a solid.

Parameters

- **sections** (*Vertex, Face, Sketch*) – slices to loft into object. If not provided, pending_faces will be used. If vertices are to be used, a vertex can be the first, last, or first and last elements.
- **ruled** (*bool, optional*) – discontiguous layer tangents. Defaults to False.
- **clean** (*bool, optional*) – Remove extraneous internal structure. Defaults to True.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

make_brake_formed(*thickness: float, station_widths: float | ~collections.abc.Iterable[float], line: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.composite.Curve | None = None, side: ~build123d.build_enums.Side = <Side.LEFT>, kind: ~build123d.build_enums.Kind = <Kind.ARC>, clean: bool = True, mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Part

Create a part typically formed with a sheet metal brake from a single outline. The line parameter describes how the material is to be bent. Either a single width value or a width value at each vertex or station is provided to control the width of the end part. Note that if multiple values are provided there must be one for each vertex and that the resulting part is composed of linear segments.

Parameters

- **thickness** (*float*) – sheet metal thickness
- **station_widths** (*Union[float, Iterable[float]]*) – width of part at each vertex or a single value. Note that this width is perpendicular to the provided line/plane.
- **line** (*Union[Edge, Wire, Curve], optional*) – outline of part. Defaults to None.
- **side** (*Side, optional*) – offset direction. Defaults to Side.LEFT.
- **kind** (*Kind, optional*) – offset intersection type. Defaults to Kind.ARC.
- **clean** (*bool, optional*) – clean the resulting solid. Defaults to True.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

Raises

- **ValueError** – invalid line type
- **ValueError** – not line provided
- **ValueError** – line not suitable
- **ValueError** – incorrect # of width values

Returns
sheet metal part

Return type
Part

make_face(*edges: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.composite.Curve | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.composite.Curve] | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Sketch

Sketch Operation: make_face

Create a face from the given perimeter edges.

Parameters

- **edges** (Edge | Wire | Curve) – perimeter edges that must combine into a single closed wire. Defaults to all sketch pending edges.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

make_hull(*edges: ~build123d.topology.one_d.Edge | ~collections.abc.Iterable[~build123d.topology.one_d.Edge] | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Sketch

Sketch Operation: make_hull

Create a face from the convex hull of the given edges

Parameters

- **edges** (Edge, *optional*) – sequence of edges to hull. Defaults to all sketch pending edges.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

mirror(*objects: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.composite.Compound | ~build123d.topology.composite.Curve | ~build123d.topology.composite.Sketch | ~build123d.topology.composite.Part | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.composite.Compound | ~build123d.topology.composite.Curve | ~build123d.topology.composite.Sketch | ~build123d.topology.composite.Part] | None = None*, *about: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, -1, 0))*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Curve | Sketch | Part | Compound

Generic Operation: mirror

Applies to 1, 2, and 3 dimensional objects.

Mirror a sequence of objects over the given plane.

Parameters

- **objects** (Edge | Face | Compound *or Iterable of*) – objects to mirror
- **about** (Plane, *optional*) – reference plane. Defaults to "XZ".
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

Raises
ValueError – missing objects

offset(*objects: ~build123d.topology.one_d.Edge | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid | ~build123d.topology.composite.Compound | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid | ~build123d.topology.composite.Compound] | None = None*, *amount: float = 0*, *openings: ~build123d.topology.two_d.Face | list[~build123d.topology.two_d.Face] | None = None*, *kind: ~build123d.build_enums.Kind = <Kind.ARC>*, *side: ~build123d.build_enums.Side = <Side.BOTH>*, *closed: bool = True*, *min_edge_length: float | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.REPLACE>*) → Curve | Sketch | Part | Compound

Generic Operation: offset

Applies to 1, 2, and 3 dimensional objects.

Offset the given sequence of Edges, Faces, Compound of Faces, or Solids. The kind parameter controls the shape of the transitions. For Solid objects, the openings parameter allows selected faces to be open, like a hollow box with no lid.

Parameters

- **objects** (*Edge | Face | Solid | Compound or Iterable of*) – objects to offset
- **amount** (*float*) – positive values external, negative internal
- **openings** (*list[Face], optional*) – Defaults to None.
- **kind** (*Kind, optional*) – transition shape. Defaults to Kind.ARC.
- **side** (*Side, optional*) – side to place offset. Defaults to Side.BOTH.
- **closed** (*bool, optional*) – if Side!=BOTH, close the LEFT or RIGHT offset. Defaults to True.
- **min_edge_length** (*float, optional*) – repair degenerate edges generated by offset by eliminating edges of minimum length in offset wire. Defaults to None.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.REPLACE.

Raises

- **ValueError** – missing objects
- **ValueError** – Invalid object type

project(*objects: ~build123d.topology.one_d.Edge | ~build123d.topology.two_d.Face | ~build123d.topology.one_d.Wire | ~build123d.geometry.Vector | ~build123d.topology.zero_d.Vertex | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.two_d.Face | ~build123d.topology.one_d.Wire | ~build123d.geometry.Vector | ~build123d.topology.zero_d.Vertex] | None = None*, *workplane: ~build123d.geometry.Plane | None = None*, *target: ~build123d.topology.three_d.Solid | ~build123d.topology.composite.Compound | ~build123d.topology.composite.Part | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Curve | Sketch | Compound | ShapeList[Vector]

Generic Operation: project

Applies to 0, 1, and 2 dimensional objects.

Project the given objects or points onto a BuildLine or BuildSketch workplane in the direction of the normal of that workplane. When projecting onto a sketch a Face(s) are generated while Edges are generated for BuildLine. Will only use the first if BuildSketch has multiple active workplanes. In algebra mode a workplane must be provided and the output is either a Face, Curve, Sketch, Compound, or ShapeList[Vector].

Note that only if mode is not Mode.PRIVATE only Faces can be projected into BuildSketch and Edge/Wires into BuildLine.

Parameters

- **objects** (*Edge | Face | Wire | VectorLike | Vertex or Iterable of*) – objects or points to project
- **workplane** (*Plane, optional*) – screen workplane
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

Raises

- **ValueError** – project doesn't accept group_by
- **ValueError** – Either a workplane must be provided or a builder must be active
- **ValueError** – Points and faces can only be projected in PRIVATE mode
- **ValueError** – Edges, wires and points can only be projected in PRIVATE mode
- **RuntimeError** – BuildPart doesn't have a project operation

project_workplane(*origin: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | Vertex*, *x_dir: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | Vertex*, *projection_dir: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *distance: float*) → Plane

Part Operation: project_workplane

Return a plane to be used as a BuildSketch or BuildLine workplane with a known origin and x direction. The plane's origin will be the projection of the provided origin (in 3D space). The plane's x direction will be the projection of the provided x_dir (in 3D space).

Parameters

- **origin** (*Union[VectorLike, Vertex]*) – origin in 3D space
- **x_dir** (*Union[VectorLike, Vertex]*) – x direction in 3D space
- **projection_dir** (*VectorLike*) – projection direction
- **distance** (*float*) – distance from origin to workplane

Raises

- **RuntimeError** – Not suitable for BuildLine or BuildSketch
- **ValueError** – x_dir perpendicular to projection_dir

Returns
workplane aligned for projection

Return type
Plane

revolve(*profiles: ~build123d.topology.two_d.Face | ~collections.abc.Iterable[~build123d.topology.two_d.Face] | None = None*, *axis: ~build123d.geometry.Axis = Axis((0, 0, 0), (0, 0, 1))*, *revolution_arc: float = 360.0*, *clean: bool = True*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Part

Part Operation: Revolve

Revolve the profile or pending sketches/face about the given axis. Note that the most common use case is when the axis is in the same plane as the face to be revolved but this isn't required.

Parameters

- **profiles** (*Face, optional*) – 2D profile(s) to revolve.
- **axis** (*Axis, optional*) – axis of rotation. Defaults to Axis.Z.
- **revolution_arc** (*float, optional*) – angular size of revolution. Defaults to 360.0.
- **clean** (*bool, optional*) – Remove extraneous internal structure. Defaults to True.

- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

Raises

ValueError – Invalid axis of revolution

scale(*objects: ~build123d.topology.shape_core.Shape | ~collections.abc.Iterable[~build123d.topology.shape_core.Shape] | None = None*, *by: float | tuple[float, float, float] = 1*, *about: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None*, *mode: ~build123d.build_enums.Mode = <Mode.REPLACE>*) → Curve | Sketch | Part | Compound

Generic Operation: scale

Applies to 1, 2, and 3 dimensional objects.

Scale a sequence of objects. Note that when scaling non-uniformly across the three axes, the type of the underlying object may change to bspline from line, circle, etc.

Parameters

- **objects** (Edge | Face | Compound | Solid *or Iterable of*) – objects to scale
- **by** (*float | tuple[float, float, float]*) – scale factor
- **about** (*VectorLike, optional*) – point to scale about. Defaults to each object's location position.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.REPLACE.

Raises

ValueError – missing objects

section(*obj: ~build123d.topology.composite.Part | None = None*, *section_by: ~build123d.geometry.Plane | ~collections.abc.Iterable[~build123d.geometry.Plane] = Plane((0, 0, 0), (1, 0, 0), (0, -1, 0))*, *height: float = 0.0*, *clean: bool = True*, *mode: ~build123d.build_enums.Mode = <Mode.PRIVATE>*) → Sketch

Part Operation: section

Slices current part at the given height by section_by or current workplane(s).

Parameters

- **obj** (Part, *optional*) – object to section. Defaults to None.
- **section_by** (Plane, *optional*) – plane(s) to section object. Defaults to None.
- **height** (*float, optional*) – workplane offset. Defaults to 0.0.
- **clean** (*bool, optional*) – Remove extraneous internal structure. Defaults to True.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.INTERSECT.

split(*objects: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid] | None = None*, *bisect_by: ~build123d.geometry.Plane | ~build123d.topology.two_d.Face | ~build123d.topology.two_d.Shell = Plane((0, 0, 0), (1, 0, 0), (0, -1, 0))*, *keep: ~build123d.build_enums.Keep = <Keep.TOP>*, *mode: ~build123d.build_enums.Mode = <Mode.REPLACE>*)

Generic Operation: split

Applies to 1, 2, and 3 dimensional objects.

Bisect object with plane and keep either top, bottom or both.

Parameters

- **objects** (Edge | Wire | Face | Solid *or Iterable of*)
- **bisect_by** (Plane | Face, *optional*) – plane to segment part. Defaults to Plane.XZ.
- **keep** (Keep, *optional*) – selector for which segment to keep. Defaults to Keep.TOP.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.REPLACE.

Raises

ValueError – missing objects

sweep(*sections: ~build123d.topology.composite.Compound | ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid | ~collections.abc.Iterable[~build123d.topology.composite.Compound | ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.two_d.Face | ~build123d.topology.three_d.Solid] | None = None*, *path: ~build123d.topology.composite.Curve | ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~collections.abc.Iterable[~build123d.topology.one_d.Edge] | None = None*, *multisection: bool = False*, *is_frenet: bool = False*, *transition: ~build123d.build_enums.Transition = <Transition.TRANSFORMED>*, *normal: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None*, *binormal: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | None = None*, *clean: bool = True*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Part | Sketch

Generic Operation: sweep

Sweep pending 1D or 2D objects along path.

Parameters

- **sections** (Compound | Edge | Wire | Face | Solid) – cross sections to sweep into object
- **path** (Curve | Edge | Wire, *optional*) – path to follow. Defaults to context pending_edges.
- **multisection** (*bool*, *optional*) – sweep multiple on path. Defaults to False.
- **is_frenet** (*bool*, *optional*) – use frenet algorithm. Defaults to False.
- **transition** (Transition, *optional*) – discontinuity handling option. Defaults to Transition.TRANSFORMED.
- **normal** (*VectorLike*, *optional*) – fixed normal. Defaults to None.
- **binormal** (Edge | Wire, *optional*) – guide rotation along path. Defaults to None.
- **clean** (*bool*, *optional*) – Remove extraneous internal structure. Defaults to True.
- **mode** (Mode, *optional*) – combination. Defaults to Mode.ADD.

thicken(*to_thicken: ~build123d.topology.two_d.Face | ~build123d.topology.composite.Sketch | None = None*, *amount: float | None = None*, *normal_override: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None*, *both: bool = False*, *clean: bool = True*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Part

Part Operation: thicken

Create a solid(s) from a potentially non planar face(s) by thickening along the normals.

Parameters

- **to_thicken** (*Union[Face, Sketch]*, *optional*) – object to thicken. Defaults to None.
- **amount** (*float*) – distance to extrude, sign controls direction.

- **normal_override** (Vector, *optional*) – The normal_override vector can be used to indicate which way is 'up', potentially flipping the face normal direction such that many faces with different normals all go in the same direction (direction need only be +/- 90 degrees from the face normal). Defaults to None.
- **both** (*bool*, *optional*) – thicken in both directions. Defaults to False.
- **clean** (*bool*, *optional*) – Remove extraneous internal structure. Defaults to True.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

Raises

- **ValueError** – No object to extrude
- **ValueError** – No target object

Returns
extruded object

Return type
Part

trace(*lines: ~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.composite.Curve | ~collections.abc.Iterable[~build123d.topology.one_d.Edge | ~build123d.topology.one_d.Wire | ~build123d.topology.composite.Curve] | None = None*, *line_width: float = 1*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*) → Sketch

Sketch Operation: trace

Convert edges, wires or pending edges into faces by sweeping a perpendicular line along them.

Parameters

- **lines** (Curve | Edge | Wire | *Iterable[*Curve | Edge | Wire*]*, *optional*) – lines to trace. Defaults to sketch pending edges.
- **line_width** (*float*, *optional*) – Defaults to 1.
- **mode** (Mode, *optional*) – combination mode. Defaults to Mode.ADD.

Raises
ValueError – No objects to trace

Returns
Traced lines

Return type
Sketch

edge(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → Edge

Return Edge

Return an edge.

Parameters
select (Select, *optional*) – Edge selector. Defaults to Select.ALL.

Returns
Edge extracted

Return type
Edge

edges(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Edge]

Return Edges

Return either all or the edges created during the last operation.

Parameters
select (Select, *optional*) – Edge selector. Defaults to Select.ALL.

Returns
Edges extracted

Return type
ShapeList[Edge]

face(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → Face

Return Face

Return a face.

Parameters
select (Select, *optional*) – Face selector. Defaults to Select.ALL.

Returns
Face extracted

Return type
Face

faces(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Face]

Return Faces

Return either all or the faces created during the last operation.

Parameters
select (Select, *optional*) – Face selector. Defaults to Select.ALL.

Returns
Faces extracted

Return type
ShapeList[Face]

solid(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → Solid

Return Solid

Return a solid.

Parameters
select (Select, *optional*) – Solid selector. Defaults to Select.ALL.

Returns
Solid extracted

Return type
Solid

solids(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Solid]

Return Solids

Return either all or the solids created during the last operation.

Parameters
select (Select, *optional*) – Solid selector. Defaults to Select.ALL.

Returns
Solids extracted

Return type
ShapeList[Solid]

vertex(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → Vertex

Return Vertex

Return a vertex.

Parameters
select (Select, *optional*) – Vertex selector. Defaults to Select.ALL.

Returns
Vertex extracted

Return type
Vertex

vertices(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Vertex]

Return Vertices

Return either all or the vertices created during the last operation.

Parameters
select (Select, *optional*) – Vertex selector. Defaults to Select.ALL.

Returns
Vertices extracted

Return type
ShapeList[Vertex]

wire(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → Wire

Return Wire

Return a wire.

Parameters
select (Select, *optional*) – Wire selector. Defaults to Select.ALL.

Returns
Wire extracted

Return type
Wire

wires(*self*, *select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Wire]

Return Wires

Return either all or the wires created during the last operation.

Parameters
select (Select, *optional*) – Wire selector. Defaults to Select.ALL.

Returns
Wires extracted

Return type
ShapeList[Wire]

1.12 Topology Selection and Exploration

Topology is the structure of build123d geometric features and traversing the topology of a part is often required to specify objects for an operation or to locate a CAD feature. *Selectors* allow selection of topology objects into a *ShapeList*. *Operators* are powerful methods further explore and refine a *ShapeList* for subsequent operations.

1.12.1 Selectors

Selectors provide methods to extract all or a subset of a feature type in the referenced object. These methods select Edges, Faces, Solids, Vertices, or Wires in Builder objects or from Shape objects themselves. All of these methods return a *ShapeList*, which is a subclass of `list` and may be sorted, grouped, or filtered by *Operators*.

Overview

Selector	Criteria	Applicability	Description
vertices()	ALL, LAST	BuildLine, BuildSketch, BuildPart	Vertex extraction
edges()	ALL, LAST, NEW	BuildLine, BuildSketch, BuildPart	Edge extraction
wires()	ALL, LAST	BuildLine, BuildSketch, BuildPart	Wire extraction
faces()	ALL, LAST	BuildSketch, BuildPart	Face extraction
solids()	ALL, LAST	BuildPart	Solid extraction

Both shape objects and builder objects have access to selector methods to select all of a feature as long as they can contain the feature being selected.

```
# In context
with BuildSketch() as context:
    Rectangle(1, 1)
    context.edges()

    # Build context implicitly has access to the selector
    edges()

# Taking the sketch out of context
context.sketch.edges()

# Create sketch out of context
Rectangle(1, 1).edges()
```

Select In Build Context

Build contexts track the last operation and their selector methods can take *Select* as criteria to specify a subset of features to extract. By default, a selector will select `ALL` of a feature, while `LAST` selects features created or altered by the most recent operation. *edges()* can uniquely specify `NEW` to only select edges created in the last operation which neither existed in the referenced object before the last operation, nor the modifying object.

Important

Select as selector criteria is only valid for builder objects!

```
# In context
with BuildPart() as context:
    Box(2, 2, 1)
```

```
    Cylinder(1, 2)
    context.edges(Select.LAST)

# Does not work out of context!
context.part.edges(Select.LAST)
(Box(2, 2, 1) + Cylinder(1, 2)).edges(Select.LAST)
```

Create a simple part to demonstrate selectors. Select using the default criteria `Select.ALL`. Specifying `Select.ALL` for the selector is not required.

```
with BuildPart() as part:
    Box(5, 5, 1)
    Cylinder(1, 5)

    part.vertices()
    part.edges()
    part.faces()

    # Is the same as
    part.vertices(Select.ALL)
    part.edges(Select.ALL)
    part.faces(Select.ALL)
```

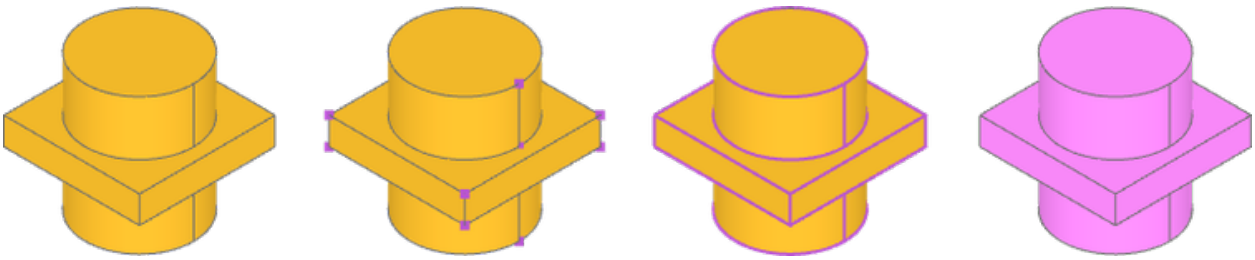


Fig. 1: The default `Select.ALL` features

Select features changed in the last operation with criteria `Select.LAST`.

```
with BuildPart() as part:
    Box(5, 5, 1)
    Cylinder(1, 5)

    part.vertices(Select.LAST)
    part.edges(Select.LAST)
    part.faces(Select.LAST)
```

Select only new edges from the last operation with `Select.NEW`. This option is only available for a `ShapeList` of edges!

```
with BuildPart() as part:
    Box(5, 5, 1)
    Cylinder(1, 5)

    part.edges(Select.NEW)
```

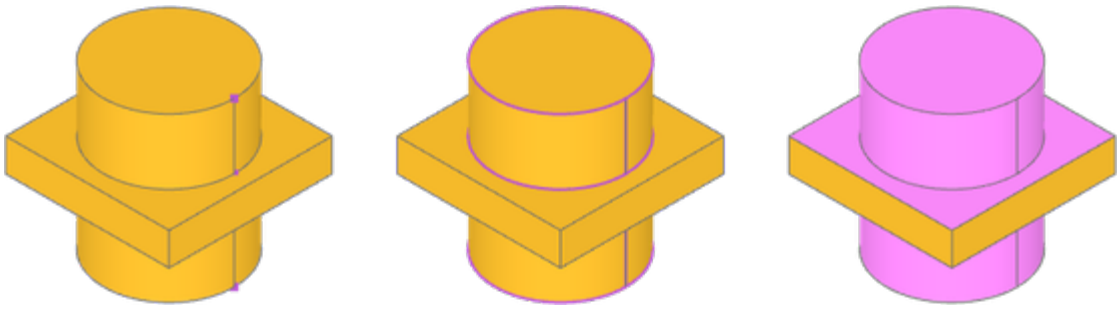


Fig. 2: `Select.LAST` features

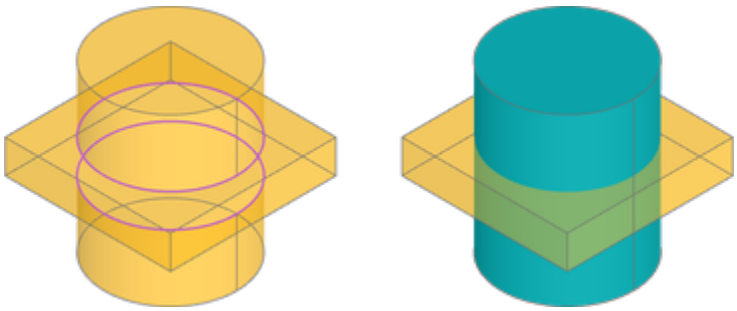


Fig. 3: `Select.NEW` edges where box and cylinder intersect

This only returns new edges which are not reused from Box or Cylinder, in this case where the objects *intersect*. But what happens if the objects don't intersect and all the edges are reused?

```
with BuildPart() as part:
    Box(5, 5, 1, align=(Align.CENTER, Align.CENTER, Align.MAX))
    Cylinder(2, 2, align=(Align.CENTER, Align.CENTER, Align.MIN))

    part.edges(Select.NEW)
```

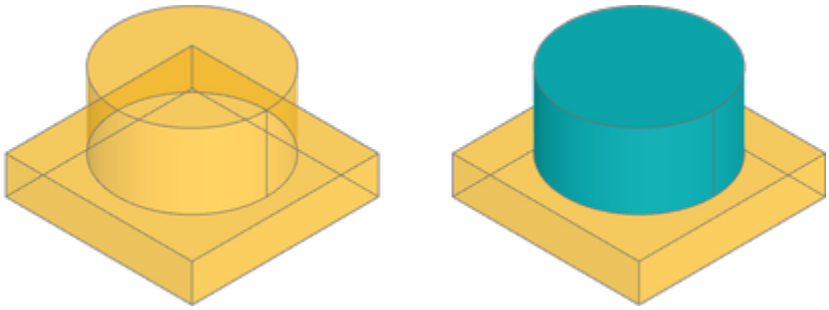


Fig. 4: `Select.NEW` edges when box and cylinder don't intersect

No edges are selected! Unlike the previous example, the Edge between the Box and Cylinder objects is an edge reused from the Cylinder. Think of `Select.NEW` as a way to select only completely new edges created by the operation.

Note

Chamfer and fillet modify the current object, but do not have new edges via `Select.NEW`.

```
with BuildPart() as part:
    Box(5, 5, 1)
    Cylinder(1, 5)
    edges = part.edges().filter_by(lambda a: a.length == 1)
    fillet(edges, 1)

    part.edges(Select.NEW)
```

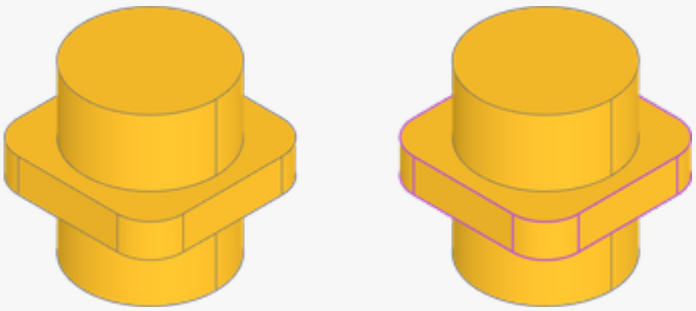
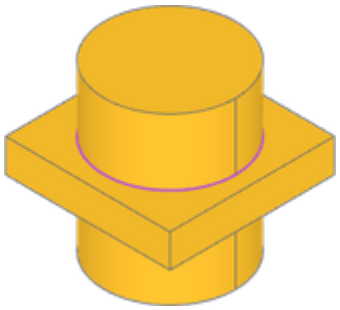


Fig. 5: Left, `Select.NEW` returns no edges after fillet. Right, `Select.LAST`

Select New Edges In Algebra Mode

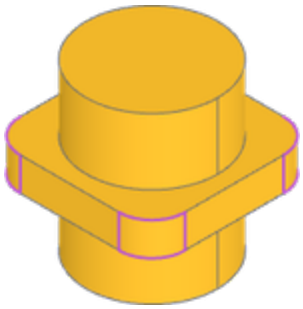
The utility method `new_edges` compares one or more shape objects to a another "combined" shape object and returns the edges new to the combined shape. `new_edges` is available both Algebra mode or Builder mode, but is necessary in Algebra Mode where `Select.NEW` is unavailable

```
box = Box(5, 5, 1)
circle = Cylinder(2, 5)
part = box + circle
edges = new_edges(box, circle, combined=part)
```



`new_edges` can also find edges created during a chamfer or fillet operation by comparing the object before the operation to the "combined" object.

```
box = Box(5, 5, 1)
circle = Cylinder(2, 5)
part_before = box + circle
edges = part_before.edges().filter_by(lambda a: a.length == 1)
part = fillet(edges, 1)
edges = new_edges(part_before, combined=part)
```



1.12.2 Operators

Operators provide methods refine a `ShapeList` of features isolated by a *selector* to further specify feature(s). These methods can sort, group, or filter `ShapeList` objects and return a modified `ShapeList`, or in the case of *group_by()*, `GroupBy`, a list of `ShapeList` objects accessible by index or key.

Overview

Method	Criteria	Description
sort_by()	`Axis`, `Edge`, `Wire`, `SortBy`, callable, property	Sort `ShapeList` by criteria
sort_by_distance()	`Shape`, `VectorLike`	Sort `ShapeList` by distance from criteria
group_by()	`Axis`, `Edge`, `Wire`, `SortBy`, callable, property	Group `ShapeList` by criteria
filter_by()	`Axis`, `Plane`, `GeomType`, callable, property	Filter `ShapeList` by criteria
filter_by_position(	`Axis`	Filter `ShapeList` by `Axis` & mix / max values

Operator methods take criteria to refine `ShapeList`. Broadly speaking, the criteria fall into the following categories, though not all operators take all criteria:

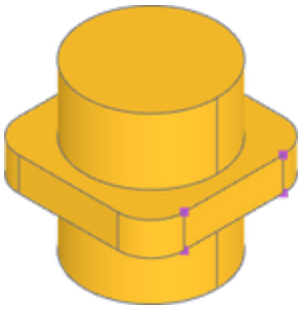
- Geometric objects: `Axis`, `Plane`
- Topological objects: `Edge`, `Wire`
- Enums: *SortBy*, *GeomType*
- Properties, eg: `Face.area`, `Edge.length`
- Callable, eg: `lambda e: e.is_interior == 1`, `lambda f: len(f.edges()) >= 3`, `Vertex().distance`, `topo_distance_to()`

Sort

A `ShapeList` can be sorted with the *sort_by()* and *sort_by_distance()* methods based on a sorting criteria. Sorting is a critical step when isolating individual features as a `ShapeList` from a selector is typically unordered.

Here we want to capture some vertices from the object furthest along `X`: All the vertices are first captured with the *vertices()* selector, then sort by `Axis.X`. Finally, the vertices can be captured with a list slice for the last 4 list items, as the items are sorted from least to greatest `X` position. Remember, `ShapeList` is a subclass of `list`, so any list slice can be used.

```
part.vertices().sort_by(Axis.X)[-4:]
```



Examples

Sort Examples

SortBy

SortBy enums are shape property shorthands which work across Shape multiple object types. SortBy is a criteria for both sort_by and group_by.

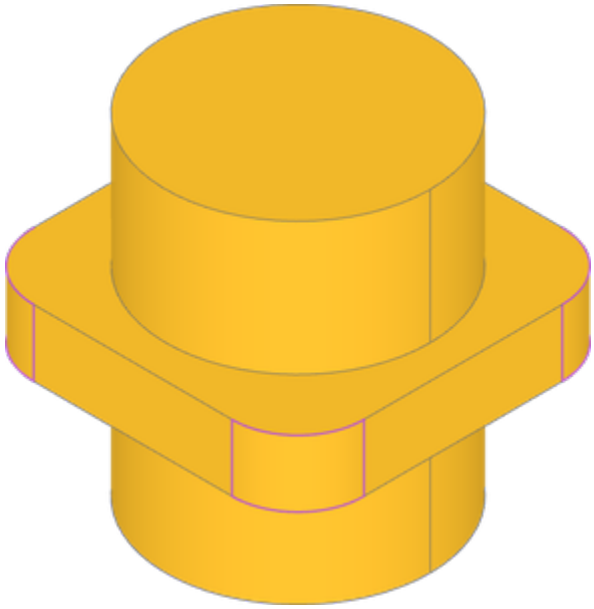
- SortBy.LENGTH works with Edge, Wire
- SortBy.AREA works with Face, Solid
- SortBy.VOLUME works with Solid
- SortBy.RADIUS works with Edge, Face with *GeomType* CIRCLE, CYLINDER, SPHERE
- SortBy.DISTANCE works Vertex, Edge, Wire, Face, Solid

SortBy is often interchangeable with specific shape properties and can alternatively be used with``group_by``.

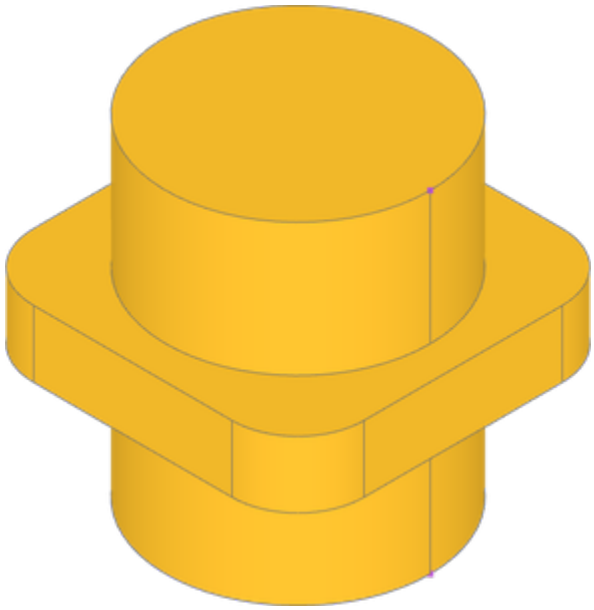
Setup

```
from build123d import *

with BuildPart() as part:
    Box(5, 5, 1)
    Cylinder(2, 5)
    edges = part.edges().filter_by(lambda a: a.length == 1)
    fillet(edges, 1)
```

```
part.wires().sort_by(SortBy.LENGTH)[:4]

part.wires().sort_by(Wire.length)[:4]
part.wires().group_by(SortBy.LENGTH)[0]
```



```
part.vertices().sort_by(SortBy.DISTANCE)[-2:]

part.vertices().sort_by_distance(Vertex())[-2:]
part.vertices().group_by(Vertex().distance)[-1]
```



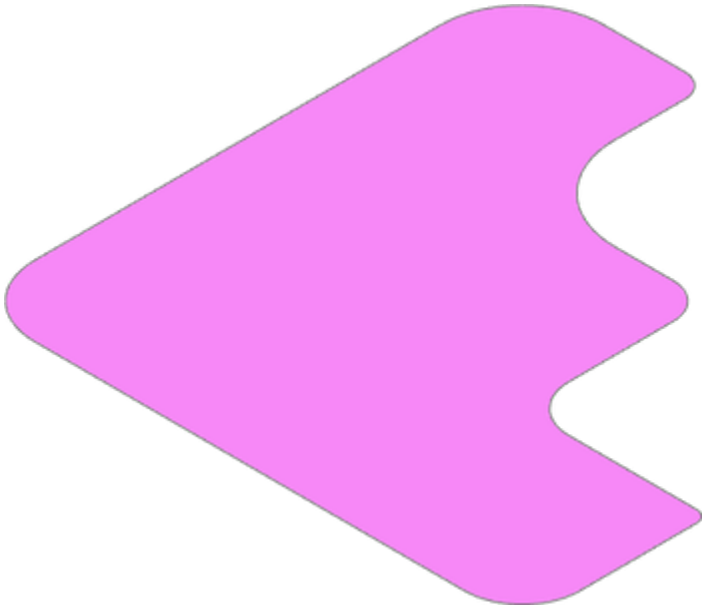
Along Wire

Vertices selected from an edge or wire might have a useful ordering when created from a single object, but when created from multiple objects, the ordering not useful. For example, when applying incrementing fillet radii to a list of vertices from the face, the order is random.

Setup

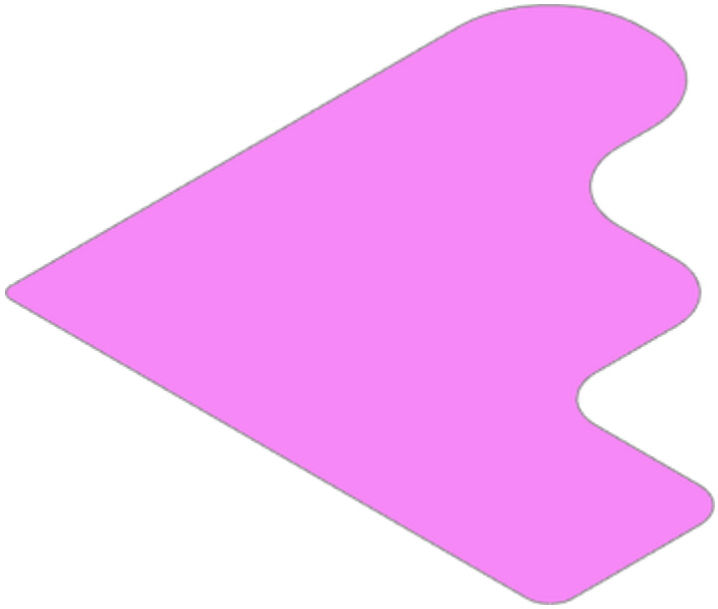
```
from build123d import *

with BuildSketch() as along_wire:
    Rectangle(48, 16, align=Align.MIN)
    Rectangle(16, 48, align=Align.MIN)
    Rectangle(32, 32, align=Align.MIN)
```

```
    for i, v in enumerate(along_wire.vertices()):
        fillet(v, i + 1)
```



Vertices may be sorted along the wire they fall on to create order. Notice the fillet radii now increase in order.

```
    sorted_verts = along_wire.vertices().sort_by(along_wire.wire())
    for i, v in enumerate(sorted_verts):
        fillet(v, i + 1)
```



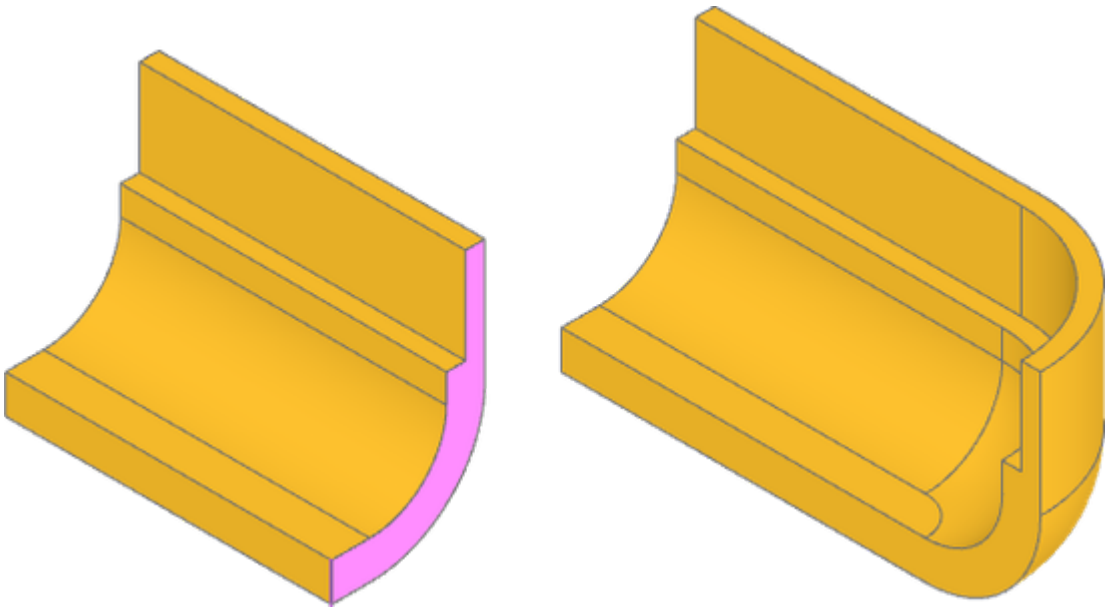
Axis

Sorting by axis is often the most straightforward way to optimize selections. In this part we want to revolve the face at the end around an inside edge of the completed extrusion. First, the face to extrude can be found by sorting along x-axis and the revolution edge can be found sorting along y-axis.

Setup

```
from build123d import *

with BuildPart() as part:
    with BuildSketch(Plane.YZ) as profile:
        with BuildLine():
            l1 = FilletPolyline((16, 0), (32, 0), (32, 25), radius=12)
            l2 = FilletPolyline((16, 4), (28, 4), (28, 15), radius=8)
            Line(l1 @ 0, l2 @ 0)
            Polyline(l1 @ 1, l1 @ 1 - Vector(2, 0), l2 @ 1 + Vector(2, 0), l2 @ 1)
        make_face()
    extrude(amount=34)
```

```
    face = part.faces().sort_by(Axis.X)[-1]
    edge = face.edges().sort_by(Axis.Y)[0]
    revolve(face, -Axis(edge), 90)
```



Distance From

A `sort_by_distance` can be used to sort objects by their distance from another object. Here we are sorting the boxes by distance from the origin, using an empty `Vertex` (at the origin) as the reference shape to find distance to.

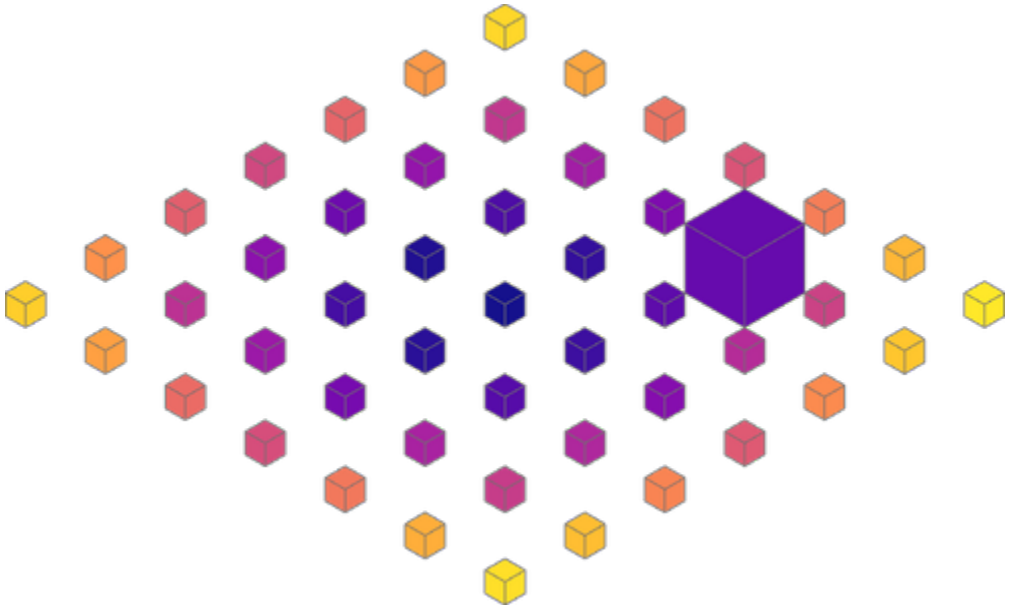
Setup

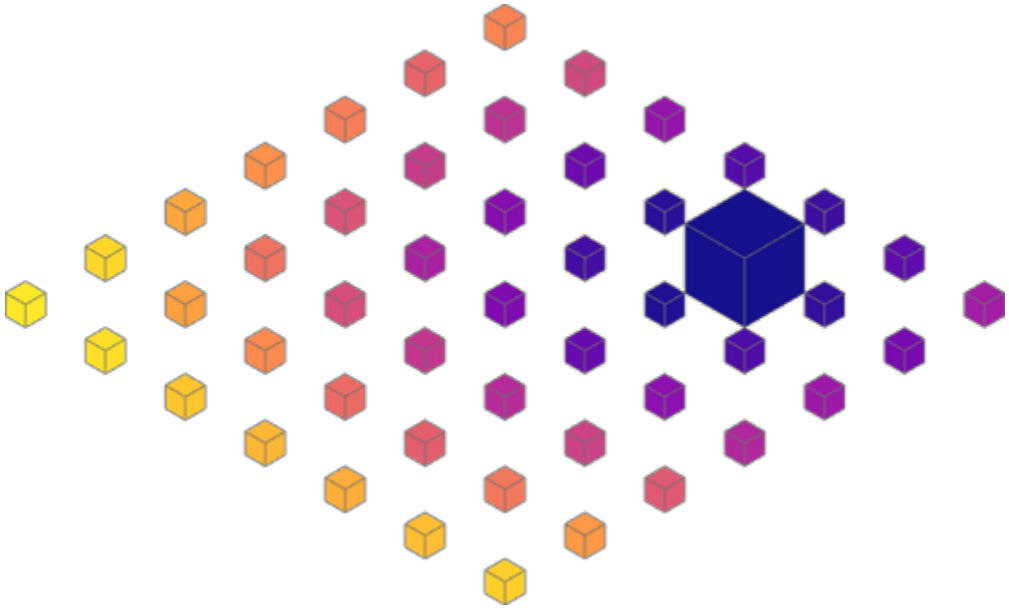
```
from itertools import product

from build123d import *
from ocp_vscode import *

boxes = ShapeList(
    Box(1, 1, 1).scale(0.75 if (i, j) == (1, 2) else 0.25).translate((i, j, 0))
    for i, j in product(range(-3, 4), repeat=2)
)
```

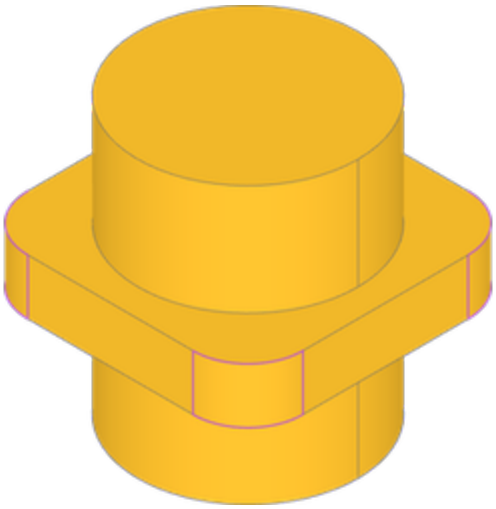
```
boxes = boxes.sort_by_distance(Vertex())
show(*boxes, colors=ColorMap.listed(len(boxes)))
```

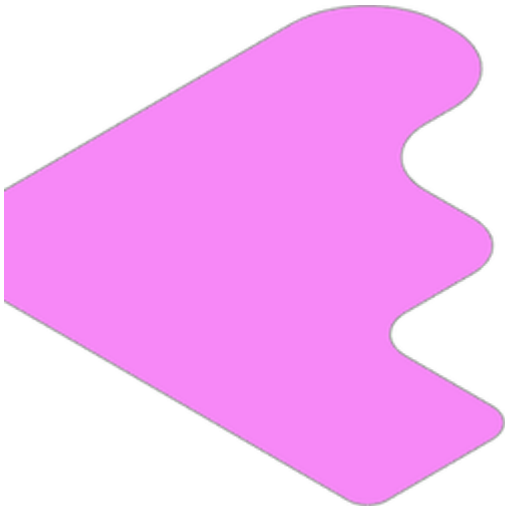
The example can be extended by first sorting the boxes by volume using the `Solid` property `volume`, and getting the last (largest) box. Then, the boxes sorted by their distance from the largest box.

```
boxes = boxes.sort_by_distance(boxes.sort_by(Solid.volume).last)
show(*boxes, colors=ColorMap.listed(len(boxes)))
```

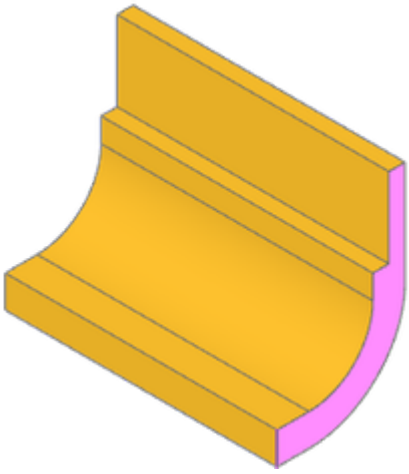




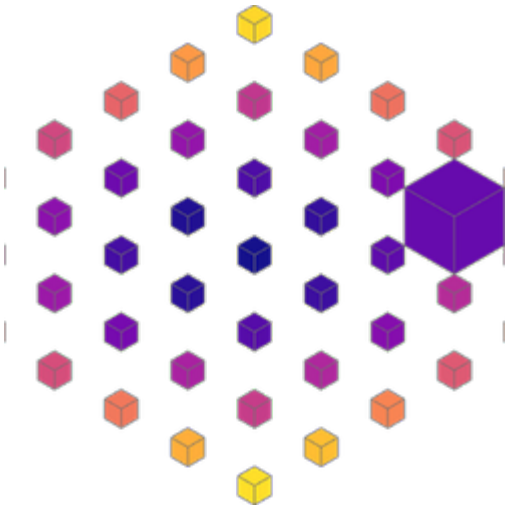




SortBy *SortBy*



Along Wire *Along Wire*



Axis *Axis*

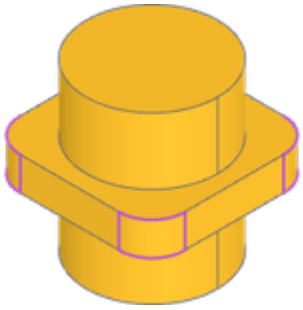
Distance From *Distance From*

Group

A ShapeList can be grouped and sorted with the *group_by()* method based on a grouping criteria. Grouping can be a great way to organize features without knowing the values of specific feature properties. Rather than returning a `ShapeList`, *group_by()* returns a `GroupBy`, a list of `ShapeList` objects sorted by the grouping criteria. `GroupBy` can be printed to view the members of each group, indexed like a list to retrieve a `ShapeList`, and be accessed using a key with the `group` method. If the group keys are unknown they can be discovered with `key_to_group_index`.

If we want only the edges from the smallest faces by area we can get the faces, then group by `SortBy.AREA`. The `ShapeList` of smallest faces is available from the first list index. Finally, a `ShapeList` has access to selectors, so calling *edges()* will return a new list of all edges in the previous list.

```
part.faces().group_by(SortBy.AREA)[0].edges())
```



Topological Distance

`topo_distance_to()` creates a callable key that measures graph distance through topology rather than geometric distance through space. It is useful when selecting features by adjacency, for example faces connected to a reference face, or the next ring of faces after that.

Distances are measured within the shared `topo_parent` of the reference shape. The reference shape has distance `0`, directly adjacent shapes have distance 1, and unreachable shapes have distance `inf`.

```
box = Box(1, 1, 1)
faces = box.faces()
top_face = faces.sort_by(Axis.Z)[-1]

face_rings = faces.group_by(topo_distance_to(top_face))

top = face_rings[0]
sides = face_rings[1]
bottom = face_rings[2]
```

Multiple reference shapes can be provided. This is useful for selecting all features within a topological distance from any reference. In this example, a sphere is converted to a triangular mesh, faces near the middle of the mesh are used as references, and all mesh faces are grouped into topological rings expanding away from that starting band.

```
from build123d import *
from pathlib import Path
from tempfile import TemporaryDirectory

from ocp_vscode import ColorMap, show

mesher = Mesher()
mesher.add_shape(Sphere(1), linear_deflection=0.05, angular_deflection=1)

with TemporaryDirectory() as tmp_dir:
    mesh_path = Path(tmp_dir) / "sphere.stl"
    mesher.write(mesh_path)
    mesh_sphere = Mesher().read(mesh_path)[0]

sphere_faces = mesh_sphere.faces()

vertical_groups = sphere_faces.group_by(Axis.Z)
starting_ring = vertical_groups[len(vertical_groups) // 2]
face_rings = sphere_faces.group_by(topo_distance_to(starting_ring))

show(*face_rings, colors=ColorMap.listed(len(face_rings)))
```

The same approach can be used with edges or vertices. For example, a single edge on the mesh can be used as the starting point for edge-distance rings.

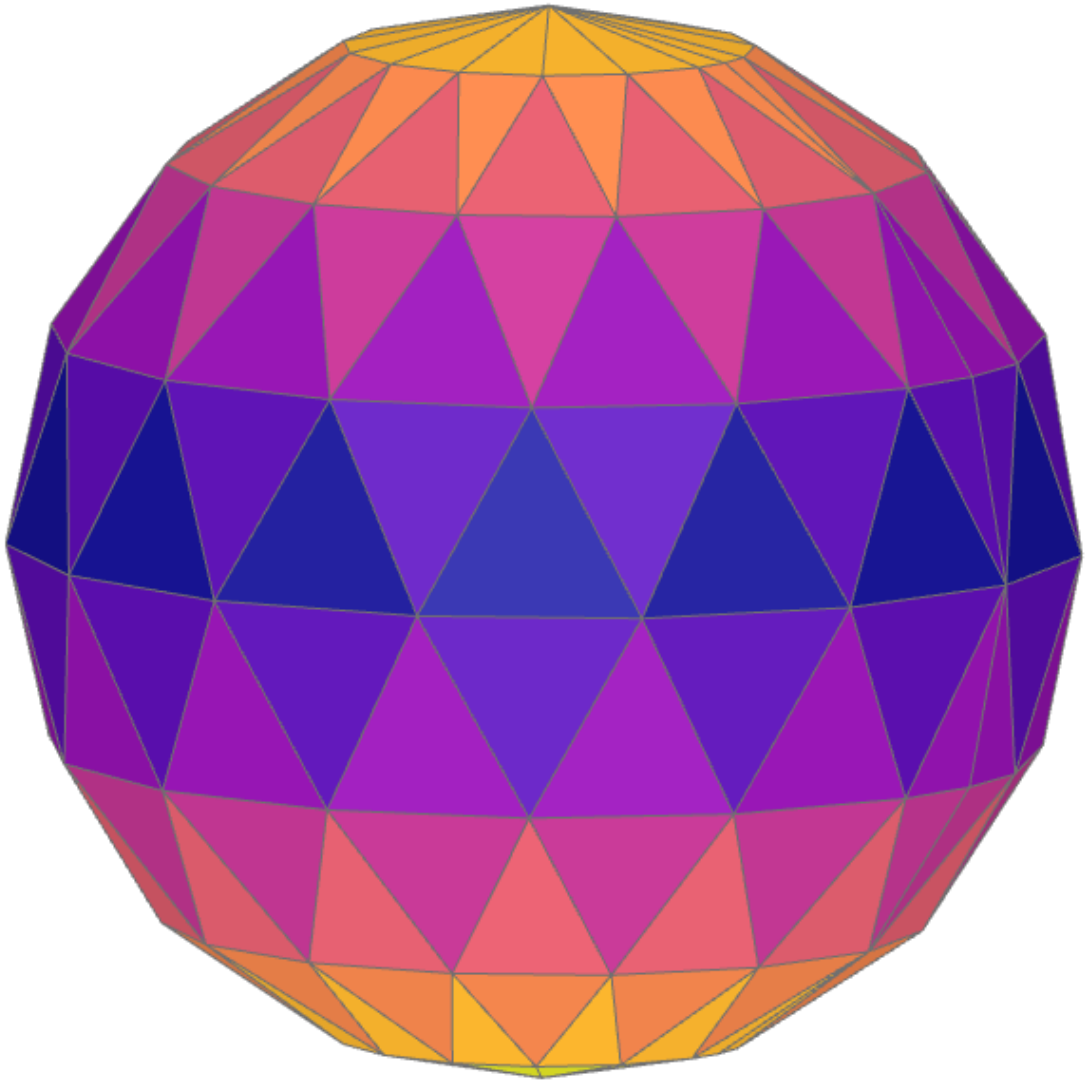
```
sphere_edges = mesh_sphere.edges()
reference_edge = choice(sphere_edges)
edge_rings = sphere_edges.group_by(topo_distance_to(reference_edge))
```

Examples

Group Examples

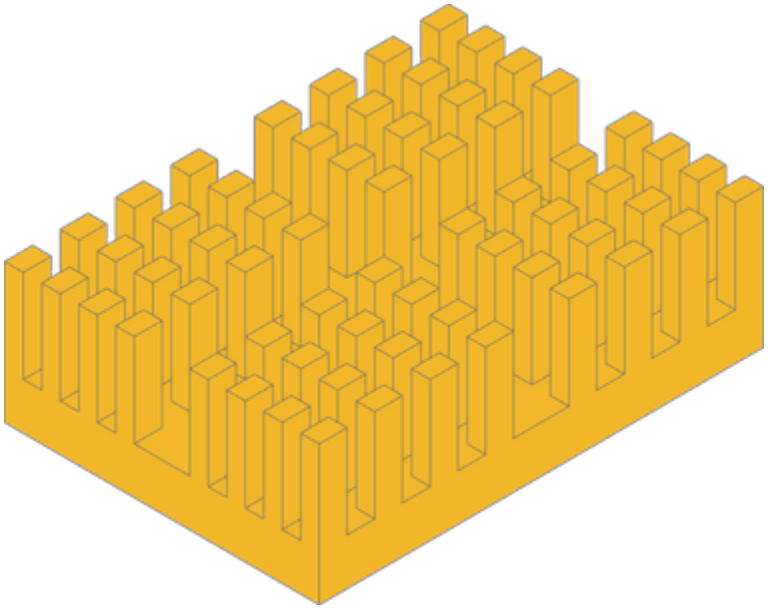
Axis and Length

This heatsink component could use fillets on the ends of the fins on the long ends. One way to accomplish this is to filter by length, sort by axis, and slice the result knowing how many edges to expect.



Setup

```
from build123d import *

with BuildPart() as fins:
    with GridLocations(4, 6, 4, 4):
        Box(2, 3, 10, align=(Align.CENTER, Align.CENTER, Align.MIN))

with BuildPart() as part:
    Box(34, 48, 5, align=(Align.CENTER, Align.CENTER, Align.MAX))
    with GridLocations(20, 27, 2, 2):
        add(fins)
```

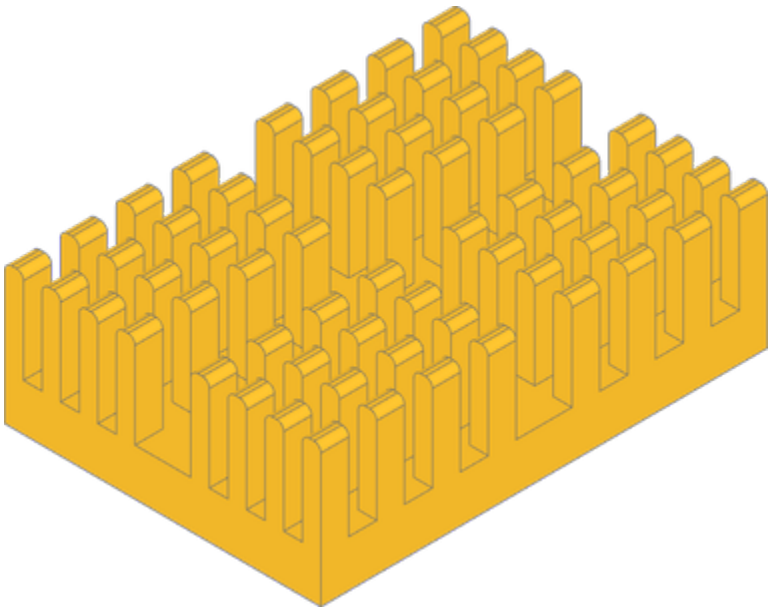


However, `group_by` can be used to first group all the edges by z-axis position and then group again by length. In both cases, you can select the desired edges from the last group.

```
    target = part.edges().group_by(Axis.Z)[-1].group_by(Edge.length)[-1]
    fillet(target, .75)
```

Hole Area

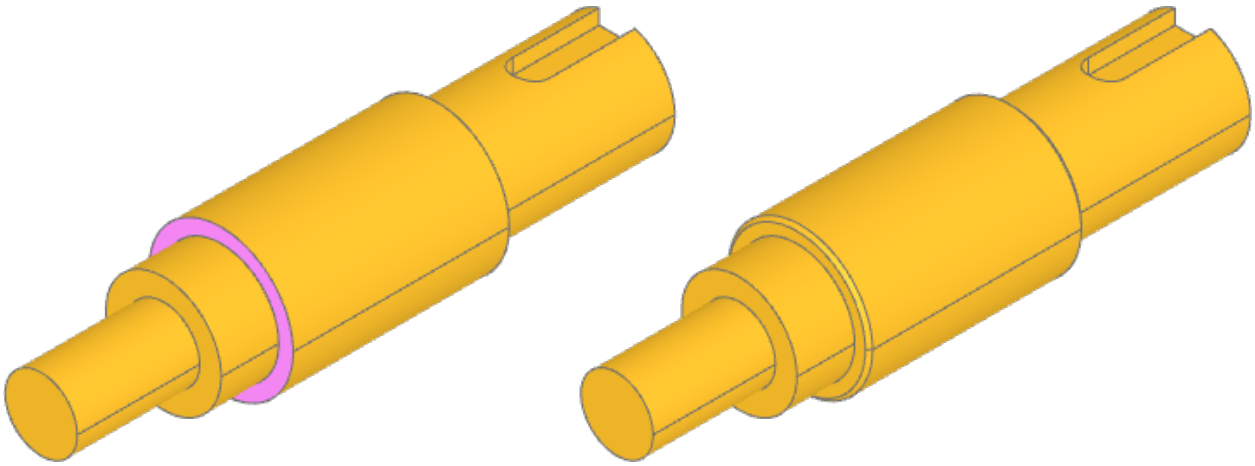
Callables are available to `group_by`, like `sort_by`. Here, the first inner wire is converted to a face and then that area is the grouping criteria to find the faces with the largest hole.



Setup

```
from build123d import *

with BuildPart() as part:
    Cylinder(10, 30, rotation=(90, 0, 0))
    Cylinder(8, 40, rotation=(90, 0, 0), align=(Align.CENTER, Align.CENTER, Align.MAX))
    Cylinder(8, 23, rotation=(90, 0, 0), align=(Align.CENTER, Align.CENTER, Align.MIN))
    Cylinder(5, 40, rotation=(90, 0, 0), align=(Align.CENTER, Align.CENTER, Align.MIN))
    with BuildSketch(Plane.XY.offset(8)) as s:
        SlotCenterPoint((0, 38), (0, 48), 5)
    extrude(amount=2.5, both=True, mode=Mode.SUBTRACT)
```

```
    faces = part.faces().group_by(
        lambda f: Face(f.inner_wires()[0]).area if f.inner_wires() else 0
    )
    chamfer([f.outer_wire().edges() for f in faces[-1]], 0.5)
```



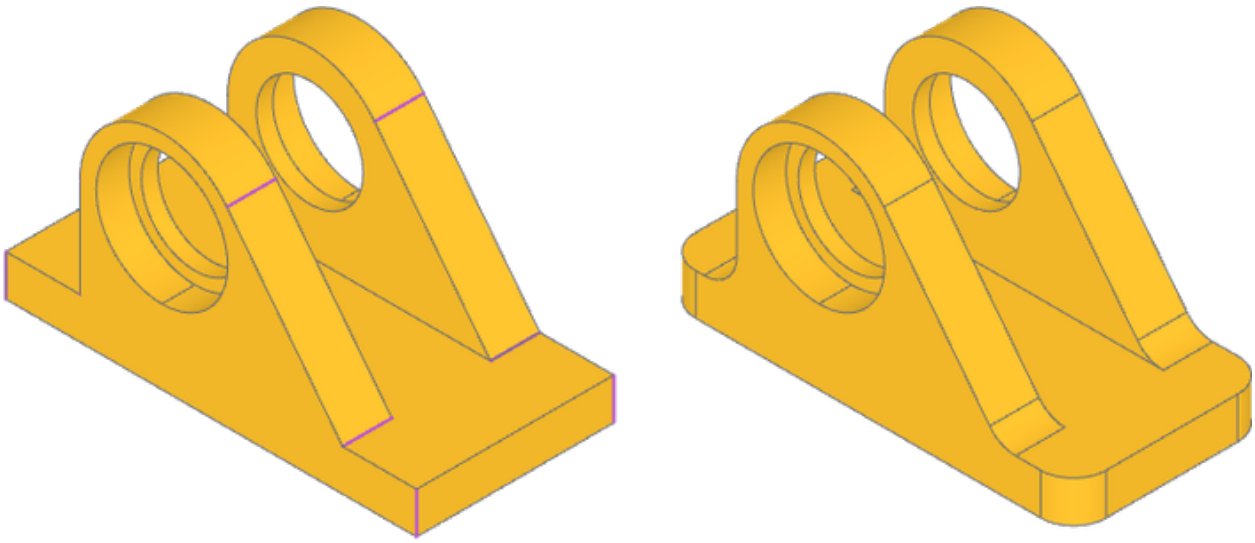
Properties with Keys

Groups are usually selected by list slice, often smallest `[0]` or largest `[-1]`, but they can also be selected by key with the `group` method if the keys are known. Starting with an incomplete bearing block we are looking to add fillets to the ribs and corners. We know the edge lengths so the edges can be grouped by `Edge.Length` and then the desired groups are selected with the `group` method using the lengths as keys.

Setup

```
from build123d import *

with BuildPart() as part:
    with BuildSketch(Plane.XZ) as sketch:
        with BuildLine():
            CenterArc((-6, 12), 10, 0, 360)
            Line((-16, 0), (16, 0))
        make_hull()
        Rectangle(50, 5, align=(Align.CENTER, Align.MAX))

    extrude(amount=12)

    Box(38, 6, 22, align=(Align.CENTER, Align.MAX, Align.MIN), mode=Mode.SUBTRACT)

    circle = part.edges().filter_by(GeomType.CIRCLE).sort_by(Axis.Y)[0]
    with Locations(Plane(circle.arc_center, z_dir=circle.normal())):
        CounterBoreHole(13 / 2, 16 / 2, 4)

    mirror(about=Plane.XZ)
```

```
    length_groups = part.edges().group_by(Edge.length)
    fillet(length_groups.group(6) + length_groups.group(5), 4)
```

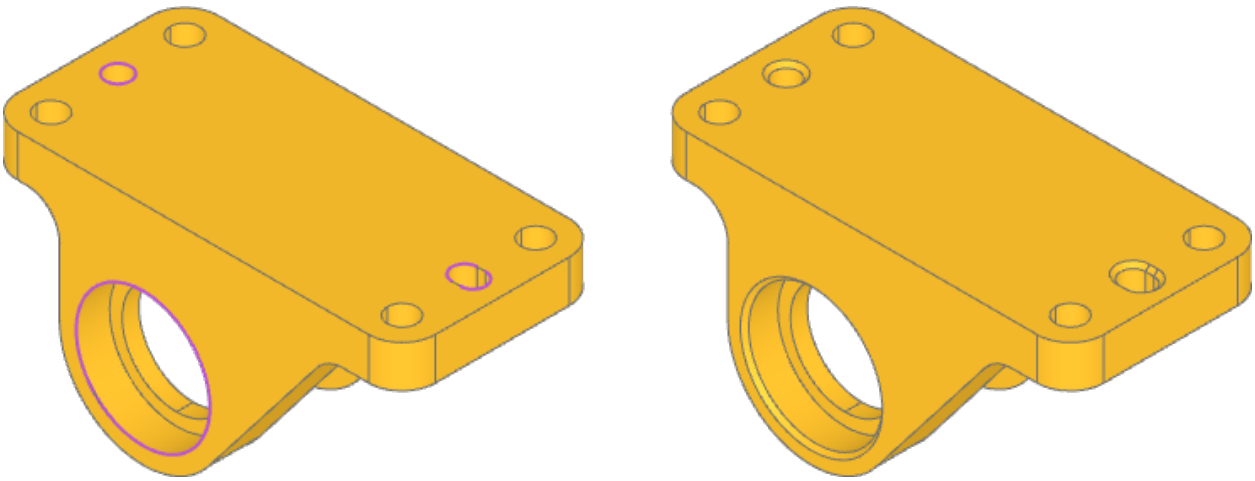


Next, we add alignment pin and counterbore holes after the fillets to make sure screw heads sit flush where they overlap the fillet. Once that is done, it's time to finalize the tight-tolerance bearing and pin holes with chamfers to make installation easier. We can filter by `GeomType.CIRCLE` and group by `Edge.radius` to group the circular edges. Again, the radii are known, so we can retrieve those groups directly and then further specify only the edges the bearings and pins are installed from.

Adding holes

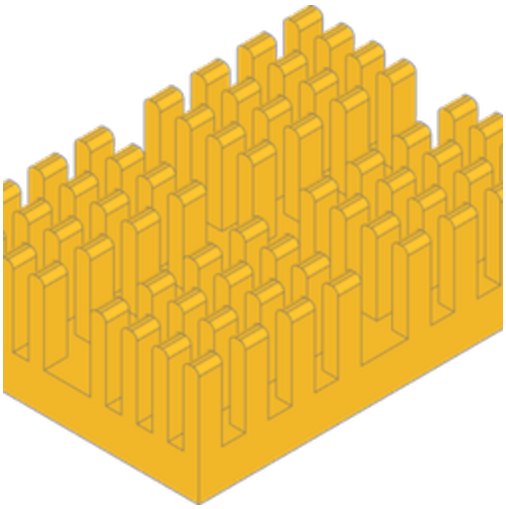
```
    with BuildSketch() as pins:
        with Locations((-21, 0)):
            Circle(3 / 2)
        with Locations((21, 0)):
            SlotCenterToCenter(1, 3)
    extrude(amount=-12, mode=Mode.SUBTRACT)

    with GridLocations(42, 16, 2, 2):
        CounterBoreHole(3.5 / 2, 3.5, 0)
```

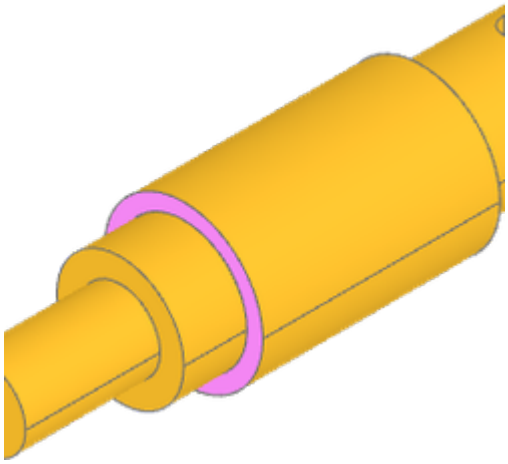
```
    radius_groups = part.edges().filter_by(GeomType.CIRCLE).group_by(Edge.radius)
    bearing_edges = radius_groups.group(8).group_by(SortBy.DISTANCE)[-1]
    pin_edges = radius_groups.group(1.5).filter_by_position(Axis.Z, -5, -5)
    chamfer([pin_edges, bearing_edges], .5)
```



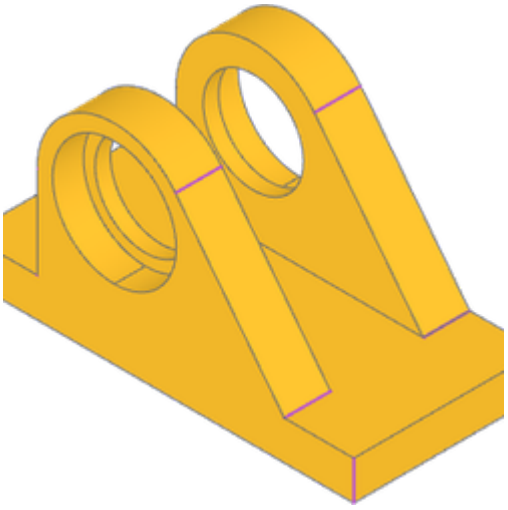
Note that `group_by` is not the only way to capture edges with a known property value! `filter_by` with a lambda expression can be used as well:

```
radius_groups = part.edges().filter_by(GeomType.CIRCLE)
bearing_edges = radius_groups.filter_by(lambda e: e.radius == 8)
pin_edges = radius_groups.filter_by(lambda e: e.radius == 1.5)
```





Axis and Length *Axis and Length*



Hole Area *Hole Area*

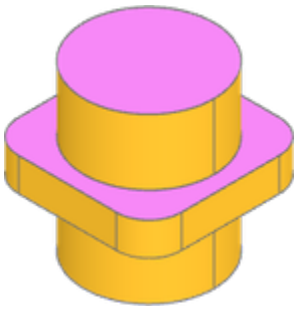
Properties with Keys *Properties with Keys*

Filter

A ShapeList can be filtered with the *filter_by()* and *filter_by_position()* methods based on a filtering criteria. Filters are flexible way to isolate (or exclude) features based on known criteria.

Lets say we need all the faces with a normal in the +Z direction. One way to do this might be with a list comprehension, however *filter_by()* has the capability to take a lambda function as a filter condition on the entire list. In this case, the normal of each face can be checked against a vector direction and filtered accordingly.

```
part.faces().filter_by(lambda f: f.normal_at() == Vector(0, 0, 1))
```



Examples

Filter Examples

GeomType

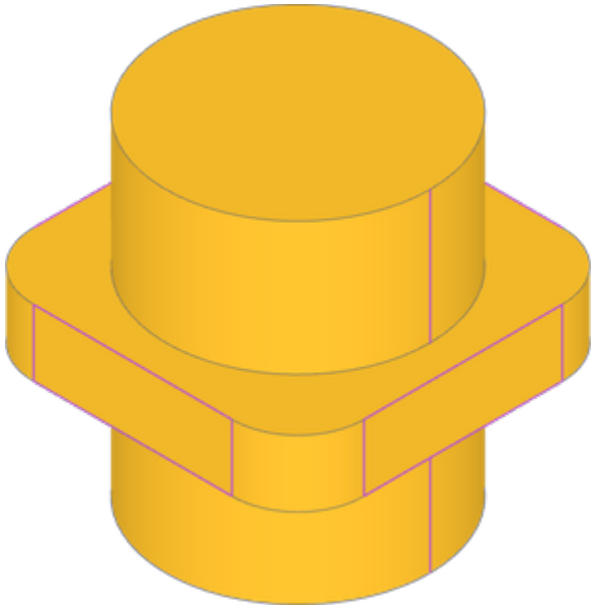
GeomType enums are shape type shorthands for Edge and Face objects. They are most helpful for filtering objects of that specific type for further operations, and are sometimes necessary e.g. before sorting or filtering by radius. Edge and Face each support a subset of GeomType:

- Edge can be type LINE, CIRCLE, ELLIPSE, HYPERBOLA, PARABOLA, BEZIER, BSPLINE, OFFSET, OTHER
- Face can be type PLANE, CYLINDER, CONE, SPHERE, TORUS, BEZIER, BSPLINE, REVOLUTION, EXTRUSION, OFFSET, OTHER

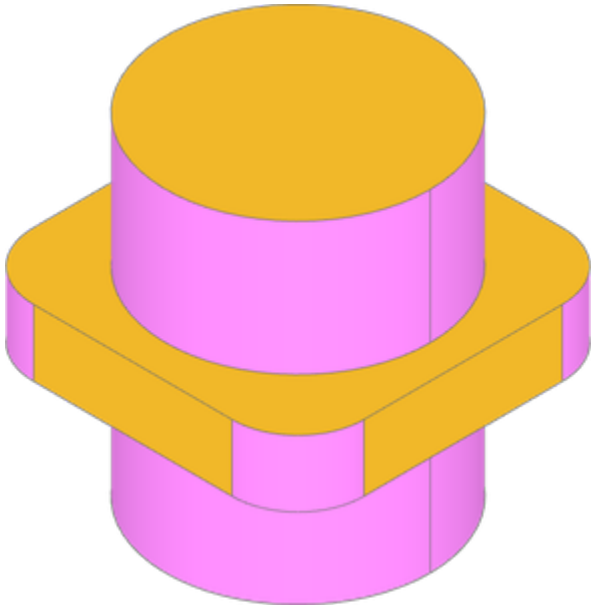
Setup

```
from build123d import *

with BuildPart() as part:
    Box(5, 5, 1)
    Cylinder(2, 5)
    edges = part.edges().filter_by(lambda a: a.length == 1)
    fillet(edges, 1)
```

```
part.edges().filter_by(GeomType.LINE)
```



```
part.faces().filter_by(GeomType.CYLINDER)
```



All Edges Circle

In this complete bearing block, we want to add joints for the bearings. These should be located in the counterbore recess. One way to locate the joints is by finding faces with centers located where the joints need to be located. Filtering for faces with only circular edges selects the counterbore faces that meet the joint criteria.

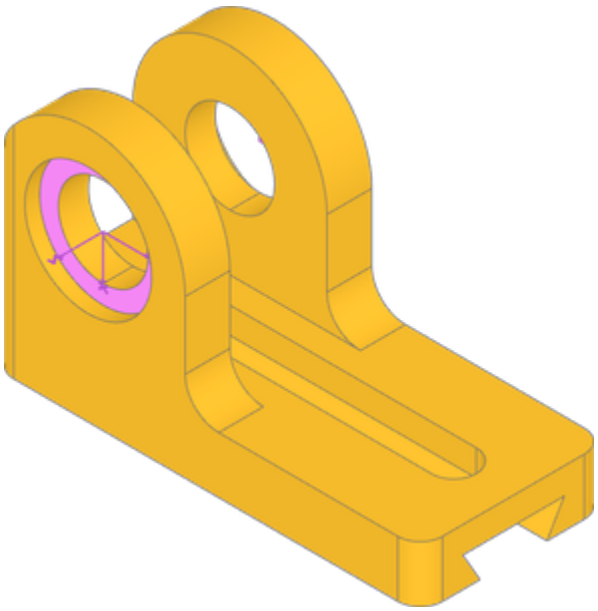
Setup

```
from build123d import *

with BuildPart() as part:
    with BuildSketch() as s:
        Rectangle(115, 50)
        with Locations((5 / 2, 0)):
            SlotOverall(90, 12, mode=Mode.SUBTRACT)
    extrude(amount=15)

    with BuildSketch(Plane.XZ.offset(50 / 2)) as s3:
        with Locations((-115 / 2 + 26, 15)):
            SlotOverall(42 + 2 * 26 + 12, 2 * 26, rotation=90)
    zz = extrude(amount=-12)
    split(bisect_by=Plane.XY)
    edgs = part.part.edges().filter_by(Axis.Y).group_by(Axis.X)[-2]
    fillet(edgs, 9)

    with Locations(zz.faces().sort_by(Axis.Y)[0]):
        with Locations((42 / 2 + 6, 0)):
            CounterBoreHole(24 / 2, 34 / 2, 4)
    mirror(about=Plane.XZ)

    with BuildSketch() as s4:
        RectangleRounded(115, 50, 6)
    extrude(amount=80, mode=Mode.INTERSECT)
    # fillet does not work right, mode intersect is safer

    with BuildSketch(Plane.YZ) as s4:
        with BuildLine() as bl:
            l1 = Line((0, 0), (18 / 2, 0))
            l2 = PolarLine(l1 @ 1, 8, 60, length_mode=LengthMode.VERTICAL)
            l3 = Line(l2 @ 1, (0, 8))
            mirror(about=Plane.YZ)
        make_face()
    extrude(amount=115 / 2, both=True, mode=Mode.SUBTRACT)
```

```
    faces = part.faces().filter_by(
        lambda f: all(e.geom_type == GeomType.CIRCLE for e in f.edges())
    )
    for i, f in enumerate(faces):
        RigidJoint(f"bearing_bore_{i}", joint_location=f.center_location)
```

Axis and Plane

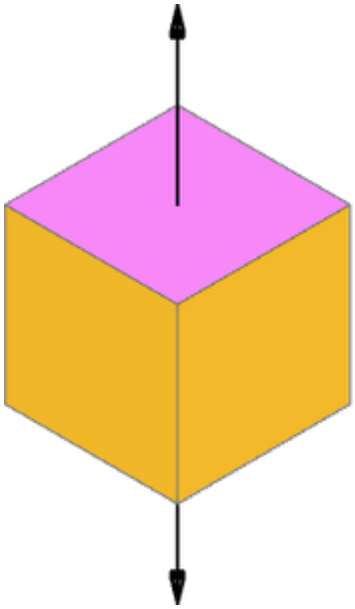
Filtering by an Axis will select faces perpendicular to the axis. Likewise filtering by Plane will select faces parallel to the plane.

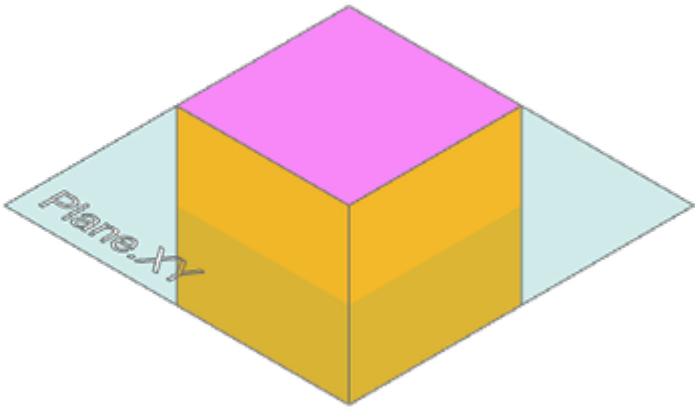


Setup

```
from build123d import *

with BuildPart() as part:
    Box(1, 1, 1)
```

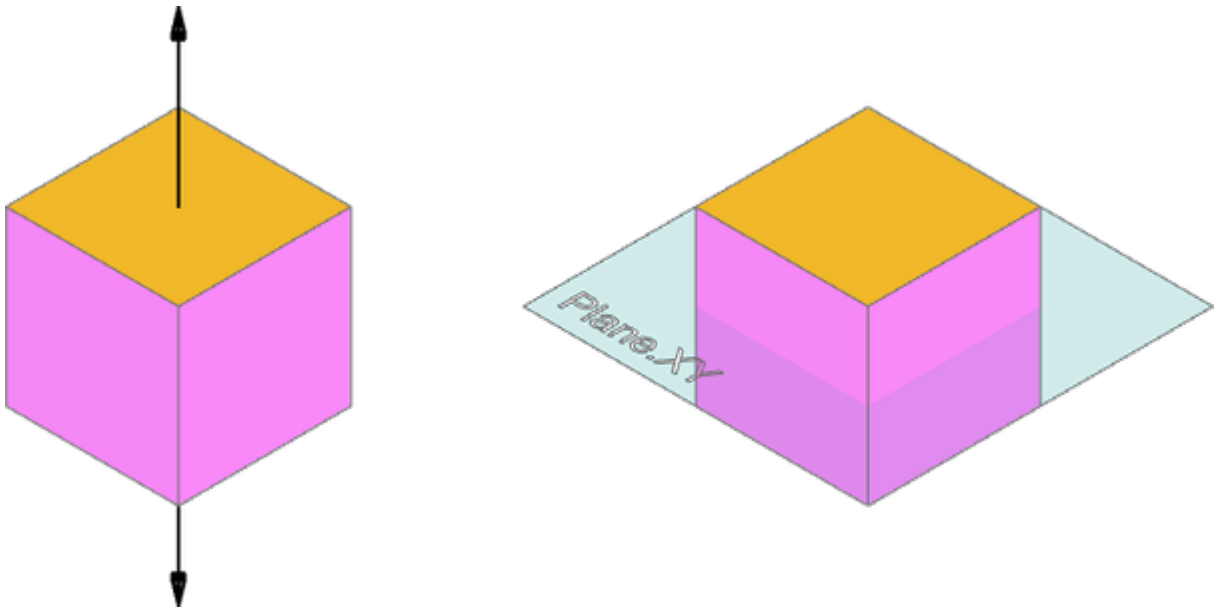
```
part.faces().filter_by(Axis.Z)
part.faces().filter_by(Plane.XY)
```





It might be useful to filter by an Axis or Plane in other ways. A lambda can be used to accomplish this with feature properties or methods. Here, we are looking for faces where the dot product of face normal and either the axis direction or the plane normal is about to 0. The result is faces parallel to the axis or perpendicular to the plane.

```
part.faces().filter_by(lambda f: abs(f.normal_at().dot(Axis.Z.direction) < 1e-6)
part.faces().filter_by(lambda f: abs(f.normal_at().dot(Plane.XY.z_dir)) < 1e-6)
```



Inner Wire Count

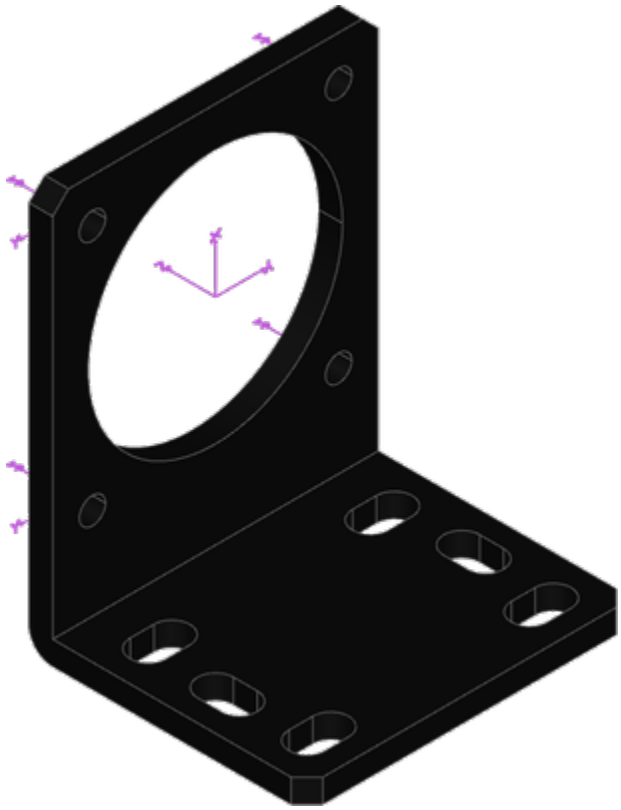
This motor bracket imported from a step file needs joints for adding to an assembly. Joints for the M3 clearance holes were already found by using the cylindrical face's axis of rotation, but the motor bore and slots need specific placement. The motor bore can be found by filtering for faces with 5 inner wires, sorting for the desired face, and then filtering for the specific inner wire by radius.

- bracket STEP model: `nema-17-bracket.step`

Setup

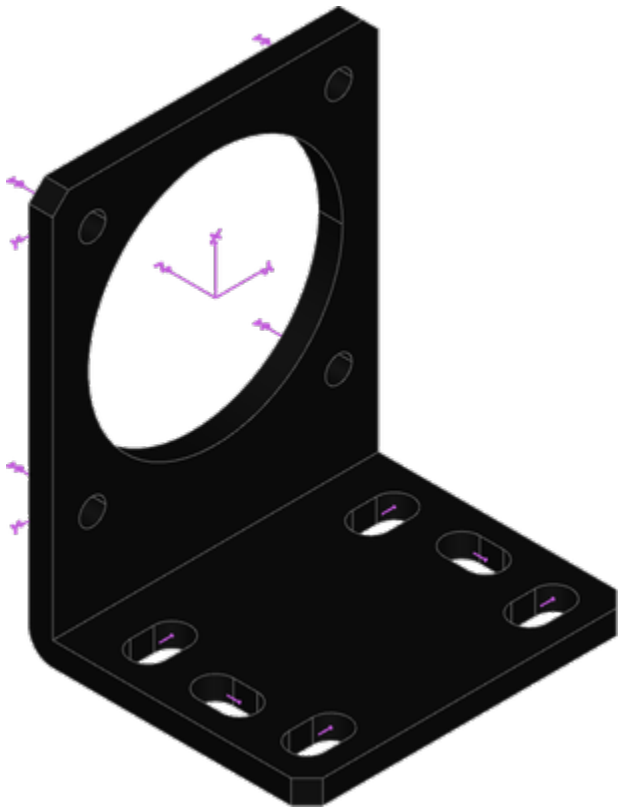
```
from build123d import *

bracket = import_step(os.path.join(working_path, "nema-17-bracket.step"))
faces = bracket.faces()

motor_mounts = faces.filter_by(GeomType.CYLINDER).filter_by(lambda f: f.radius == 3.3/2)
for i, f in enumerate(motor_mounts):
    location = f.axis_of_rotation.location
    RigidJoint(f"motor_m3_{i}", bracket, joint_location=location)
```

```
motor_face = faces.filter_by(lambda f: len(f.inner_wires()) == 5).sort_by(Axis.X)[-1]
motor_bore = motor_face.inner_wires().edges().filter_by(lambda e: e.radius == 16).edge()
location = Location(motor_bore.arc_center, motor_bore.normal() * 90, Intrinsic.YXZ)
RigidJoint(f"motor", bracket, joint_location=location)
```



Linear joints for the slots are appropriate for mating flexibility, but require more than a single location. The slot arc centers can be used for creating a linear joint axis and range. To do that we can filter for faces with 6 inner wires, sort for and select the top face, and then filter for the circular edges of the inner wires.

```
mount_face = faces.filter_by(lambda f: len(f.inner_wires()) == 6).sort_by(Axis.Z)[-1]
mount_slots = mount_face.inner_wires().edges().filter_by(GeomType.CIRCLE)
joint_edges = [
    Line(mount_slots[i].arc_center, mount_slots[i + 1].arc_center)
    for i in range(0, len(mount_slots), 2)
]
for i, e in enumerate(joint_edges):
    LinearJoint(f"mount_m4_{i}", bracket, axis=Axis(e), linear_range=(0, e.length / 2))
```



Nested Filters

Filters can be nested to specify features by characteristics other than their own, like child properties. Here we want to chamfer the mating edges of the D bore and square shaft. A way to do this is first looking for faces with only 2 line edges among the inner wires. The nested filter captures the straight edges, while the parent filter selects faces based on the count. Then, from those faces, we filter for the wires with any line edges.

Setup

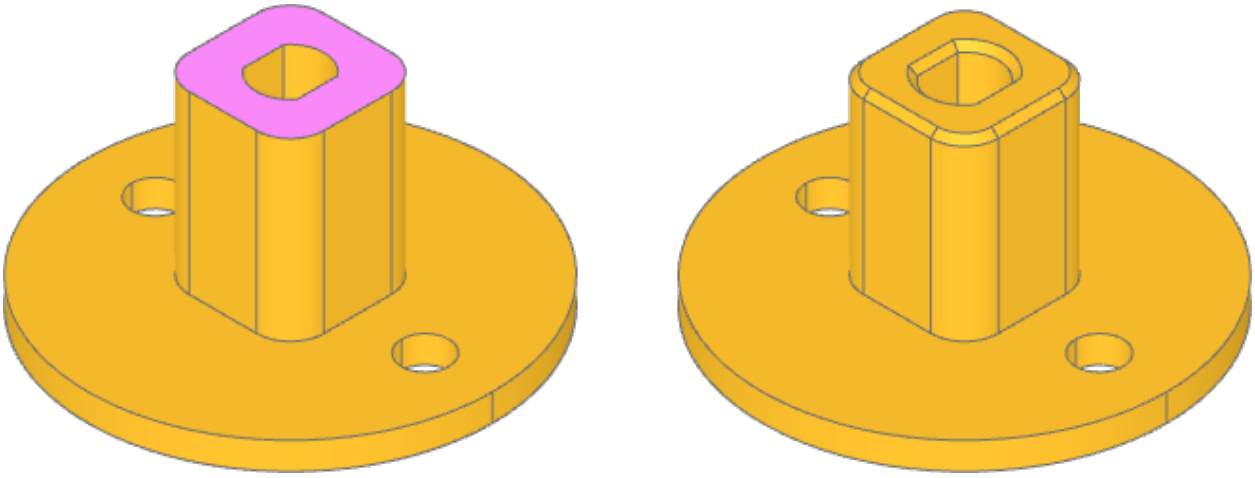
```
from build123d import *

with BuildPart() as part:
    Cylinder(15, 2, align=(Align.CENTER, Align.CENTER, Align.MIN))
    with BuildSketch():
        RectangleRounded(10, 10, 2.5)
    extrude(amount=15)

    with BuildSketch():
        Circle(2.5)
        Rectangle(4, 5, mode=Mode.INTERSECT)
    extrude(amount=15, mode=Mode.SUBTRACT)

    with GridLocations(20, 0, 2, 1):
        Hole(3.5 / 2)
```

```
    faces = part.faces().filter_by(
```

(continues on next page)

(continued from previous page)

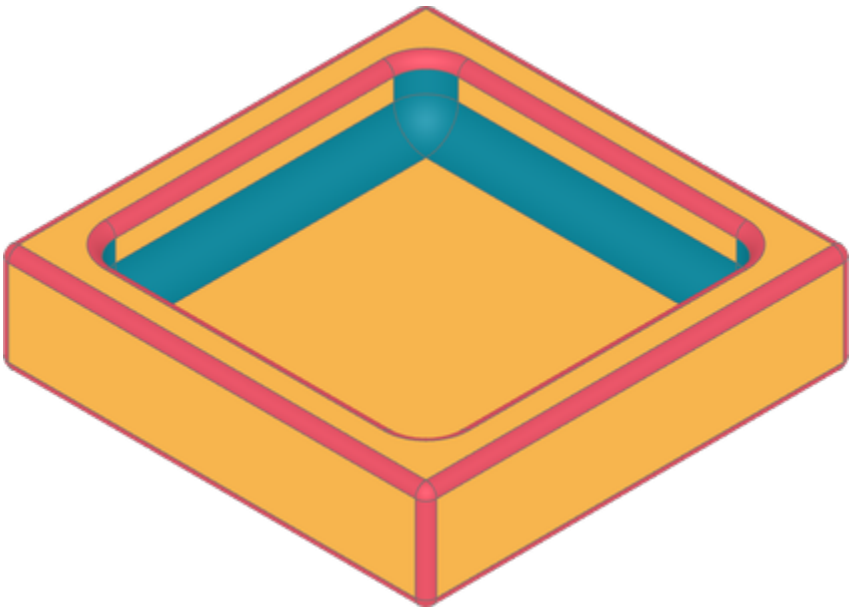
```
        lambda f: len(f.inner_wires().edges().filter_by(GeomType.LINE)) == 2
    )
    wires = faces.wires().filter_by(
        lambda w: any(e.geom_type == GeomType.LINE for e in w.edges())
    )
    chamfer(wires.edges(), 0.5)
```

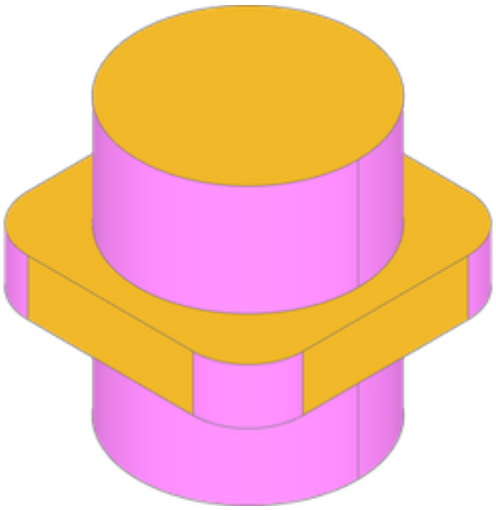


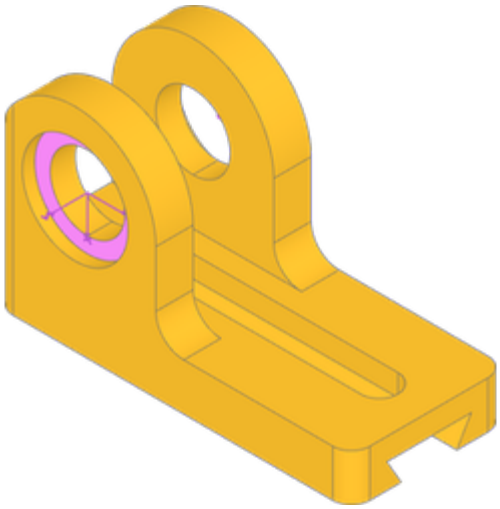
Shape Properties

Selected features can be quickly filtered by feature properties. First, we filter by interior and exterior edges using the `Edge is_interior` property to apply different fillets accordingly. Then the `Face is_circular_*` properties are used to highlight the resulting fillets.

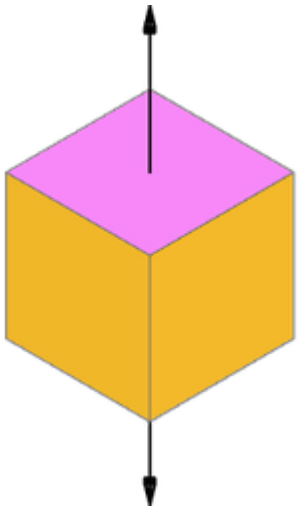
```
from build123d import *
from ocp_vscode import *

with BuildPart() as open_box_builder:
    Box(20, 20, 5)
    offset(amount=-2, openings=open_box_builder.faces().sort_by(Axis.Z)[-1])
    inside_edges = open_box_builder.edges().filter_by(Edge.is_interior)
    fillet(inside_edges, 1.5)
    outside_edges = open_box_builder.edges().filter_by(Edge.is_interior, reverse=True)
    fillet(outside_edges, 0.5)

open_box = open_box_builder.part
open_box.color = Color(0xEDAE49)
outside_fillets = Compound(open_box.faces().filter_by(Face.is_circular_convex))
outside_fillets.color = Color(0xD1495B)
inside_fillets = Compound(open_box.faces().filter_by(Face.is_circular_concave))
inside_fillets.color = Color(0x00798C)
```



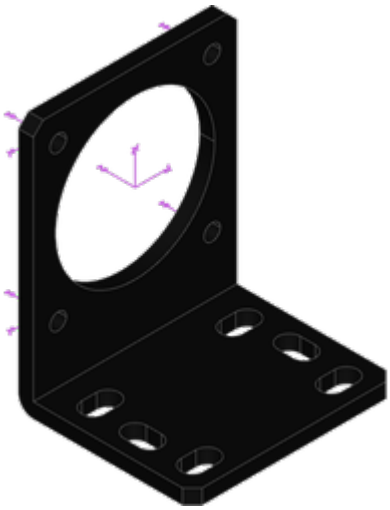




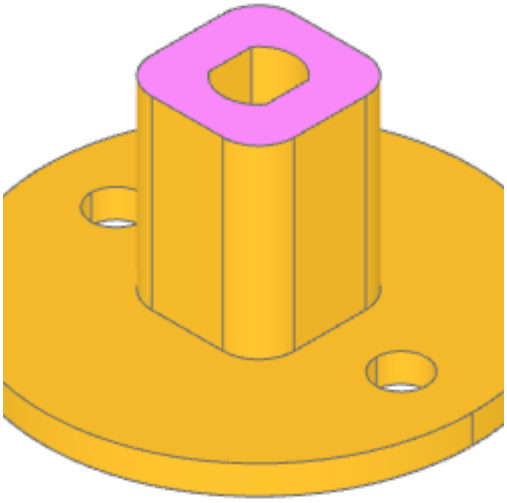
GeomType *GeomType*



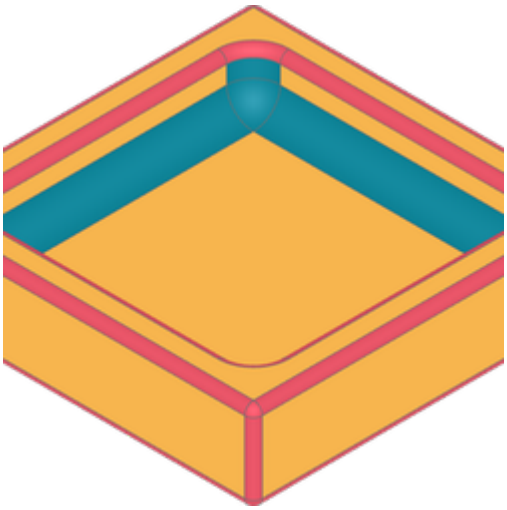
All Edges Circle *All Edges Circle*



Axis and Plane *Axis and Plane*



Inner Wire Count *Inner Wire Count*



Nested Filters *Nested Filters*

Shape Properties *Shape Properties*

1.13 Builders

The following sections describe each of the build123d stateful context builders.

1.13.1 BuildLine

BuildLine is a python context manager that is used to create one dimensional objects - objects with the property of length but not area - that are typically used as part of a BuildSketch sketch or a BuildPart path.

The complete API for BuildLine is located at the end of this section.

Basic Functionality

The following is a simple BuildLine example:

```
with BuildLine() as example_1:
    Line((0, 0), (2, 0))
    ThreePointArc((0, 0), (1, 1), (2, 0))
```

The `with` statement creates the `BuildLine` context manager with the identifier `example_1`. The objects and operations that are within the scope (i.e. indented) of this context will contribute towards the object being created by the context manager. For `BuildLine`, this object is `line` and it's referenced as `example_1.line`.

The first object in this example is a `Line` object which is used to create a straight line from coordinates (0,0) to (2,0) on the default XY plane. The second object is a `ThreePointArc` that starts and ends at the two ends of the line.

Constraints

Building with constraints enables the designer to capture design intent and add a high degree of robustness to their designs. The following sections describe creating positional and tangential constraints as well as using object attributes to enable this type of design.

@ position_at Operator

In the previous example, the `ThreePointArc` started and ended at the two ends of the `Line` but this was done by referring to the same point `(0,0)` and `(2,0)`. This can be improved upon by specifying constraints that lock the arc to those two end points, as follows:

```
with BuildLine() as example_2:
    l1 = Line((0, 0), (2, 0))
    l2 = ThreePointArc(l1 @ 0, (1, 1), l1 @ 1)
```

Here instance variables `l1` and `l2` are assigned to the two BuildLine objects and the `ThreePointArc` references the beginning of the straight line with `l1 @ 0` and the end with `l1 @ 1`. The `@` operator takes a float (or integer) parameter between 0 and 1 and determines a position at this fractional position along the line's length.

This example can be improved on further by calculating the mid-point of the arc as follows:

```
with BuildLine() as example_3:
    l1 = Line((0, 0), (2, 0))
    l2 = ThreePointArc(l1 @ 0, l1 @ 0.5 + (0, 1), l1 @ 1)
```

Here `l1 @ 0.5` finds the center of `l1` while `l1 @ 0.5 + (0, 1)` does a vector addition to generate the point `(1,1)`.

To make the design even more parametric, the height of the arc can be calculated from `l1` as follows:

```
with BuildLine() as example_4:
    l1 = Line((0, 0), (2, 0))
    l2 = ThreePointArc(l1 @ 0, l1 @ 0.5 + (0, l1.length / 2), l1 @ 1)
```

The arc height is now calculated as `(0, l1.length / 2)` by using the `length` property of `Edge` and `Wire` shapes. At this point the `ThreePointArc` is fully parametric and able to generate the same shape for any horizontal line.

% tangent_at Operator

The other operator that is commonly used within BuildLine is `%` the tangent at operator. Here is another example:

```
with BuildLine() as example_5:
    l1 = Line((0, 0), (5, 0))
    l2 = Line(l1 @ 1, l1 @ 1 + (0, l1.length - 1))
    l3 = JernArc(start=l2 @ 1, tangent=l2 % 1, radius=0.5, arc_size=90)
    l4 = Line(l3 @ 1, (0, l2.length + l3.radius))
```

which generates (note that the circles show line junctions):

The `JernArc` has the following parameters:

- `start=l2 @ 1` - start the arc at the end of line `l2`,
- `tangent=l2 % 1` - the tangent of the arc at the start point is equal to the `l2`'s, tangent at its end (shown as a dashed line)
- `radius=0.5` - the radius of the arc, and
- `arc_size=90` the angular size of the arc.

The final line starts at the end of `l3` and ends at a point calculated from the length of `l2` and the radius of arc `l3`.

Building with constraints as shown here will ensure that your designs both fully represent design intent and are robust to design changes.

BuildLine to BuildSketch

As mentioned previously, one of the two primary reasons to create BuildLine objects is to use them in BuildSketch. When a BuildLine context manager exits and is within the scope of a BuildSketch context manager it will transfer the generated line to BuildSketch. The BuildSketch *make_face()* or *make_hull()* operations are then used to transform the line (specifically a list of Edges) into a Face - the native BuildSketch objects.

Here is an example of using BuildLine to create an object that otherwise might be difficult to create:

```
with BuildSketch() as example_6:
    with BuildLine() as club_outline:
        l0 = Line((0, -188), (76, -188))
        b0 = Bezier(l0 @ 1, (61, -185), (33, -173), (17, -81))
        b1 = Bezier(b0 @ 1, (49, -128), (146, -145), (167, -67))
        b2 = Bezier(b1 @ 1, (187, 9), (94, 52), (32, 18))
        b3 = Bezier(b2 @ 1, (92, 57), (113, 188), (0, 188))
        mirror(about=Plane.YZ)
    make_face()
```

which generates:

Note

SVG import to BuildLine

The BuildLine code used in this example was generated by translating a SVG file into BuildLine source code with the *import_svg_as_buildline_code()* function. For example:

```
svg_code, builder_name = import_svg_as_buildline_code("club.svg")
```

would translate the "club.svg" image file's paths into BuildLine code much like that shown above. From there it's easy for a user to add constraints or otherwise enhance the original image and use it in their design.

BuildLine to BuildPart

The other primary reasons to use BuildLine is to create paths for BuildPart *sweep()* operations. Here some curved and straight segments define a path:

```
with BuildPart() as example_7:
    with BuildLine() as example_7_path:
        l1 = RadiusArc((0, 0), (1, 1), 2)
        l2 = Spline(l1 @ 1, (2, 3), (3, 3), tangents=(l1 % 1, (0, -1)))
        l3 = Line(l2 @ 1, (3, 0))
    with BuildSketch(Plane(origin=l1 @ 0, z_dir=l1 % 0)) as example_7_section:
        Circle(0.1)
    sweep()
```

which generates:

There are few things to note from this example:

- The @ and % operators are used to create a plane normal to the beginning of the path with which to create the circular section used by the sweep operation (this plane is not one of the ordinal planes).
- Both the path generated by BuildLine and the section generated by BuildSketch have been transferred to BuildPart when each of them exit.
- The BuildPart `Sweep` operation is using the path and section previously transferred to it (as "pending" objects) as parameters of the sweep. The `Sweep` operation "consumes" these pending objects as to not interfere with subsequence operations.

Working on other Planes

So far all of the examples were created on `Plane.XY` - the default plane - which is equivalent to global coordinates. Sometimes it's convenient to work on another plane, especially when creating paths for BuildPart `Sweep` operations.

```
with BuildLine(Plane.YZ) as example_8:
    l1 = Line((0, 0), (5, 0))
    l2 = Line(l1 @ 1, l1 @ 1 + (0, l1.length - 1))
    l3 = JernArc(start=l2 @ 1, tangent=l2 % 1, radius=0.5, arc_size=90)
    l4 = Line(l3 @ 1, (0, l2.length + l3.radius))
```

which generates:

Here the BuildLine object is created on `Plane.YZ` just by specifying the working plane during BuildLine initialization.

There are three rules to keep in mind when working with alternate planes in BuildLine:

1. `BuildLine` accepts a single `Plane` to work with as opposed to other Builders that accept more than one workplane.
2. Values entered as tuples such as `(1, 2)` or `(1, 2, 3)` will be localized to the current workplane. This rule applies to points and to the use of tuples to modify locations calculated with the `@` and `%` operators such as `l1 @ 1 + (1, 1)`. For example, if the workplane is `Plane.YZ` the local value of `(1, 2)` would be converted to `(0, 1, 2)` in global coordinates. Three tuples are converted as well - `(1, 2, 3)` on `Plane.YZ` would be `(3, 1, 2)` in global coordinates. Providing values in local coordinates allows the designer to automate such conversions.
3. Values entered using the `Vector` class or those generated by the `@` operator are considered global values and are not localized. For example: `Line(Vector(1, 2, 3), Vector(4, 5, 6))` will generate the same line

independent of the current workplane. It's unlikely that users will need to use `Vector` values but the option is there.

Finally, BuildLine's workplane need not be one of the predefined ordinal planes, it could be one created from a surface of a BuildPart part that is currently under construction.

Reference

class BuildLine(*workplane: ~build123d.topology.two_d.Face | ~build123d.geometry.Plane | ~build123d.geometry.Location = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

The BuildLine class is a subclass of Builder for building lines (objects with length but not area or volume). It has an _obj property that returns the current line being built. The class overrides the faces and solids methods of Builder since they don't apply to lines.

BuildLine only works with a single workplane which is used to convert tuples as inputs to global coordinates. For example:

```
with BuildLine(Plane.YZ) as radius_arc:
    RadiusArc((1, 2), (2, 1), 1)
```

creates an arc from global points (0, 1, 2) to (0, 2, 1). Note that points entered as Vector(x, y, z) are considered global and are not localized.

The workplane is also used to define planes parallel to the workplane that arcs are created on.

Parameters

- **workplane** (*Union[Face, Plane, Location], optional*) – plane used when local coordinates are used and when creating arcs. Defaults to Plane.XY.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

face(**args*)

face() not implemented

faces(**args*)

faces() not implemented

property line: Curve | None

Get the current line

solid(**args*)

solid() not implemented

solids(**args*)

solids() not implemented

1.13.2 BuildSketch

BuildSketch is a python context manager that is used to create planar two dimensional objects - objects with the property of area but not volume - that are typically used as profiles for BuildPart operations like *extrude()* or *revolve()*.

The complete API for BuildSketch is located at the end of this section.

Basic Functionality

The following is a simple BuildSketch example:

```
length, radius = 40.0, 60.0

with BuildSketch() as circle_with_hole:
    Circle(radius=radius)
    Rectangle(width=length, height=length, mode=Mode.SUBTRACT)
```

The `with` statement creates the `BuildSketch` context manager with the identifier `circle_with_hole`. The objects and operations that are within the scope (i.e. indented) of this context will contribute towards the object being created by the context manager. For `BuildSketch`, this object is `sketch` and it's referenced as `circle_with_hole.sketch`.

The first object in this example is a `Circle` object which is used to create a filled circular shape on the default XY plane. The second object is a `Rectangle` that is subtracted from the circle as directed by the `mode=Mode.SUBTRACT` parameter. A key aspect of sketch objects is that they are all filled shapes and not just a shape perimeter which enables combining subsequent shapes with different modes (the valid values of Mode are `ADD`, `SUBTRACT`, `INTERSECT`, `REPLACE`, and `PRIVATE`).

Sketching on other Planes

Often when designing parts one needs to build on top of other features. To facilitate doing this `BuildSketch` allows one to create sketches on any Plane while allowing the designer to work in a local X, Y coordinate system. It might be helpful to think of what is happening with this metaphor:

1. When instantiating `BuildSketch` one or more workplanes can be passed as parameters. These are the placement targets for the completed sketch.
2. The designer draws on a flat "drafting table" which is `Plane.XY`.
3. Once the sketch is complete, it's applied like a sticker to all of the workplanes passed in step 1.

As an example, let's build the following simple control box with a display on an angled plane:

Here is the code:

```
with BuildPart() as controller:
    # Create the side view of the controller
    with BuildSketch(Plane.YZ) as profile:
        with BuildLine():
            Polyline((0, 0), (0, 40), (20, 80), (40, 80), (40, 0), (0, 0))
        # Create a filled face from the perimeter drawing
        make_face()
    # Extrude to create the basis controller shape
    extrude(amount=30, both=True)
    # Round off all the edges
    fillet(controller.edges(), radius=3)
    # Hollow out the controller
    offset(amount=-1, mode=Mode.SUBTRACT)
    # Extract the face that will house the display
    display_face = (
        controller.faces()
        .filter_by(GeomType.PLANE)
```

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```
        .filter_by_position(Axis.Z, 50, 70)[0]
    )
    # Create a workplane from the face
    display_workplane = Plane(
        origin=display_face.center(), x_dir=(1, 0, 0), z_dir=display_face.normal_at()
    )
    # Place the sketch directly on the controller
    with BuildSketch(display_workplane) as display:
        RectangleRounded(40, 30, 2)
        with GridLocations(45, 35, 2, 2):
            Circle(1)
    # Cut the display sketch through the controller
    extrude(amount=-1, mode=Mode.SUBTRACT)
```

The highlighted part of the code shows how a face is extracted from the design, a workplane is constructed from this face and finally this workplane is passed to `BuildSketch` as the target for the complete sketch. Notice how the `display` sketch uses local coordinates for its features thus avoiding having the user to determine how to move and rotate the sketch to get it where it should go.

Note that `BuildSketch` accepts a sequence planes, faces and locations for workplanes so creation of an explicit workplane is often not required. Being able to work on multiple workplanes at once allows for features to be created on multiple side of an object - say both the top and bottom - which is convenient for symmetric parts.

Local vs. Global Sketches

In the above example the target for the sketch was not `Plane.XY` but a workplane passed by the user. Internally `BuildSketch` is always creating the sketch on `Plane.XY` which one can see by looking at the `sketch_local` property of your sketch. For example, to display the local version of the `display` sketch from above, one would use:

```
show_object(display.sketch_local, name="sketch on Plane.XY")
```

while the sketches as applied to their target workplanes is accessible through the `sketch` property, as follows:

```
show_object(display.sketch, name="sketch on target workplane(s)")
```

When using the *add()* operation to add an external Face to a sketch the face will automatically be reoriented to `Plane.XY` before being combined with the sketch. As Faces don't provide an x-direction it's possible that the new Face may not be oriented as expected. To reorient the Face manually to `Plane.XY` one can use the `to_local_coords()` method as follows:

```
reoriented_face = plane.to_local_coords(face)
```

where `plane` is the plane that `face` was constructed on.

Locating Features

Within a sketch features are positioned with `Locations` contexts (see *Location Context*) on the current workplane(s). The following location contexts are available within a sketch:

- *GridLocations* : a X/Y grid of locations
- *HexLocations* : a hex grid of locations ideal for nesting circles
- *Locations* : a sequence of arbitrary locations
- *PolarLocations* : locations defined by radius and angle

Generally one would specify tuples of (X, Y) values when defining locations but there are many options available to the user.

Reference

class BuildSketch(**workplanes: ~build123d.topology.two_d.Face | ~build123d.geometry.Plane | ~build123d.geometry.Location*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

The BuildSketch class is a subclass of Builder for building planar 2D sketches (objects with area but not volume) from faces or lines. It has an _obj property that returns the current sketch being built. The sketch property consists of the sketch(es) applied to the input workplanes while the sketch_local attribute is the sketch constructed on Plane.XY. The class overrides the solids method of Builder since they don't apply to lines.

Note that all sketch construction is done within sketch_local on Plane.XY. When objects are added to the sketch they must be coplanar to Plane.XY, usually handled automatically but may need user input for Edges and Wires since their construction plane isn't always able to be determined.

Parameters

- **workplanes** (*Union[Face, Plane, Location], optional*) – objects converted to plane(s) to place the sketch on. Defaults to Plane.XY.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

consolidate_edges() → Wire | list[Wire]

Unify pending edges into one or more Wires

property sketch

The global version of the sketch - may contain multiple sketches

property sketch_local: Sketch | None

Get the builder's object

solid(**args*)

solid() not implemented

solids(**args*)

solids() not implemented

1.13.3 BuildPart

BuildPart is a python context manager that is used to create three dimensional objects - objects with the property of volume - that are typically finished parts.

The complete API for BuildPart is located at the end of this section.

Basic Functionality

The following is a simple BuildPart example:

```
length, width, thickness = 80.0, 60.0, 10.0
center_hole_dia = 22.0

with BuildPart() as ex2:
    Box(length, width, thickness)
    Cylinder(radius=center_hole_dia / 2, height=thickness, mode=Mode.SUBTRACT)
```

The `with` statement creates the `BuildPart` context manager with the identifier `ex2` (this code is the second of the introductory examples). The objects and operations that are within the scope (i.e. indented) of this context will contribute towards the object being created by the context manager. For `BuildPart`, this object is `part` and it's referenced as `ex2.part`.

The first object in this example is a `Box` object which is used to create a polyhedron with rectangular faces centered on the default `Plane.XY`. The second object is a `Cylinder` that is subtracted from the box as directed by the `mode=Mode.SUBTRACT` parameter thus creating a hole.

Implicit Parameters

The BuildPart context keeps track of pending objects such that they can be used implicitly - there are a couple things to consider when deciding how to proceed:

- For sketches, the planes that they were constructed on is maintained in internal data structures such that operations like *extrude()* will have a good reference for the extrude direction. One can pass a Face to extrude but it will then be forced to use the normal direction at the center of the Face as the extrude direction which unfortunately can be reversed in some circumstances.
- Implicit parameters save some typing but hide some functionality - users have to decide what works best for them.

This tea cup example uses implicit parameters - note the *sweep()* operation on the last line:

```
from build123d import *
from ocp_vscode import show

wall_thickness = 3 * MM
fillet_radius = wall_thickness * 0.49

with BuildPart() as tea_cup:
    # Create the bowl of the cup as a revolved cross section
    with BuildSketch(Plane.XZ) as bowl_section:
        with BuildLine():
            # Start & end points with control tangents
            s = Spline(
                (30 * MM, 10 * MM),
                (69 * MM, 105 * MM),
                tangents=((1, 0.5), (0.7, 1)),
                tangent_scalars=(1.75, 1),
            )
            # Lines to finish creating ½ the bowl shape
            Polyline(s @ 0, s @ 0 + (10 * MM, -10 * MM), (0, 0), (0, (s @ 1).Y), s @ 1)
        make_face()  # Create a filled 2D shape
    revolve(axis=Axis.Z)
    # Hollow out the bowl with openings on the top and bottom
    offset(amount=-wall_thickness, openings=tea_cup.faces().filter_by(GeomType.PLANE))
    # Add a bottom to the bowl
    with Locations((0, 0, (s @ 0).Y)):
        Cylinder(radius=(s @ 0).X, height=wall_thickness)
    # Smooth out all the edges
    fillet(tea_cup.edges(), radius=fillet_radius)

    # Determine where the handle contacts the bowl
```

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```
    handle_intersections = [
        tea_cup.part.find_intersection_points(
            Axis(origin=(0, 0, vertical_offset), direction=(1, 0, 0))
        )[-1][0]
        for vertical_offset in [35 * MM, 80 * MM]
    ]
    # Create a path for handle creation
    with BuildLine(Plane.XZ) as handle_path:
        Spline(
            handle_intersections[0] - (wall_thickness / 2, 0),
            handle_intersections[0] + (35 * MM, 30 * MM),
            handle_intersections[0] + (40 * MM, 60 * MM),
            handle_intersections[1] - (wall_thickness / 2, 0),
            tangents=((1, 1.25), (-0.2, -1)),
        )
    # Align the cross section to the beginning of the path
    with BuildSketch(handle_path.line ^ 0) as handle_cross_section:
        RectangleRounded(wall_thickness, 8 * MM, fillet_radius)
    sweep()  # Sweep handle cross section along path

assert abs(tea_cup.part.volume - 130326) < 1

show(tea_cup, names=["tea cup"])
```

sweep() requires a 2D cross section - `handle_cross_section` - and a path - `handle_path` - which are both passed implicitly.

Units

Parts created with build123d have no inherent units associated with them. However, when exporting parts to external formats like `STL` or `STEP` the units are assumed to be millimeters (mm). To be more explicit with units one can use the technique shown in the above tea cup example where linear dimensions are followed by `* MM` which multiplies the dimension by the `MM` scaling factor - in this case `1`.

The following dimensional constants are pre-defined:

```
MM = 1
CM = 10 * MM
M = 1000 * MM
IN = 25.4 * MM
FT = 12 * IN
THOU = IN / 1000
```

Some export formats like DXF have the ability to explicitly set the units used.

Reference

class BuildPart(**workplanes: ~build123d.topology.two_d.Face | ~build123d.geometry.Plane | ~build123d.geometry.Location*, *mode: ~build123d.build_enums.Mode = <Mode.ADD>*)

The BuildPart class is another subclass of Builder for building parts (objects with the property of volume) from sketches or 3D objects. It has an _obj property that returns the current part being built, and several pending lists for storing faces, edges, and planes that will be integrated into the final part later. The class overrides the _add_to_pending method of Builder.

Parameters

- **workplanes** (*Plane, optional*) – initial plane to work on. Defaults to Plane.XY.
- **mode** (*Mode, optional*) – combination mode. Defaults to Mode.ADD.

property location: Location | None

Builder's location

property part: Part | None

Get the current part

property pending_edges_as_wire: Wire

Return a wire representation of the pending edges

1.14 Joints

Joint's enable Solid and Compound objects to be arranged relative to each other in an intuitive manner - with the same degree of motion that is found with the equivalent physical joints. *Joint*'s always work in pairs - a *Joint* can only be connected to another *Joint* as follows:

Joint	connect_to	Example
BallJoint	*RigidJoint*	Gimbal
CylindricalJoint	*RigidJoint*	Screw
LinearJoint	*RigidJoint*, *RevoluteJoint*	Slider or Pin Slot
RevoluteJoint	*RigidJoint*	Hinge
RigidJoint	*RigidJoint*	Fixed

Objects may have many joints bound to them each with an identifying label. All *Joint* objects have a `symbol` property that can be displayed to help visualize their position and orientation (the ocp-vscode viewer has built-in support for displaying joints).

Note

If joints are created within the scope of a *BuildPart* builder, the `to_part` parameter need not be specified as the builder will, on exit, automatically transfer the joints created in its scope to the part created.

The following sections provide more detail on the available joints and describes how they are used.

1.14.1 Rigid Joint

A rigid joint positions two components relative to each another with no freedom of movement. When a *RigidJoint* is instantiated it's assigned a `label`, a part to bind to (`to_part`), and a `joint_location` which defines both the position and orientation of the joint (see *Location*) - as follows:

```
RigidJoint(label="outlet", to_part=pipe, joint_location=path.location_at(1))
```

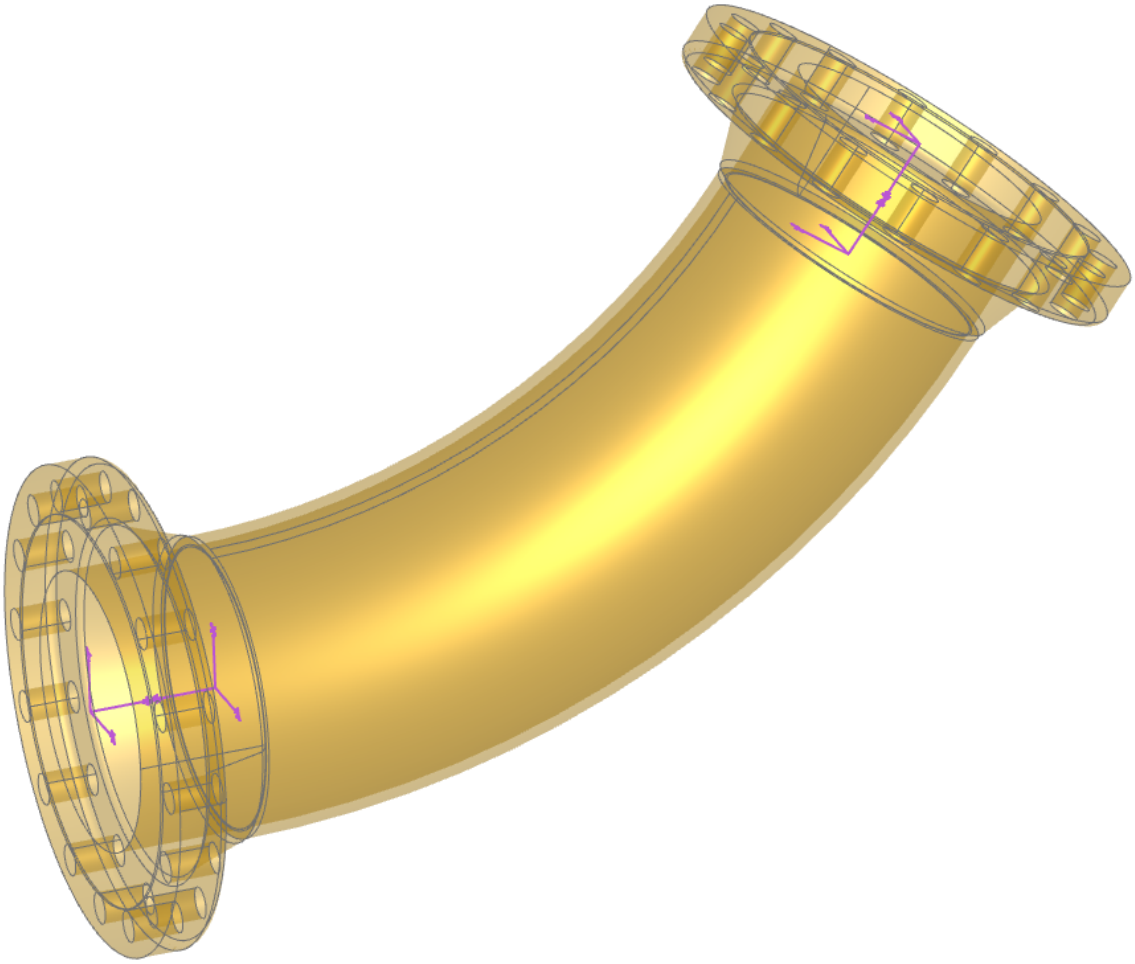
Once a joint is bound to a part this way, the *connect_to()* method can be used to repositioning another part relative to `self` which stay fixed - as follows:

```
pipe.joints["outlet"].connect_to(flange_outlet.joints["pipe"])
```

> **Note**
>
> Within a part all of the joint labels must be unique.

The *connect_to()* method only does a one time re-position of a part and does not bind them in any way; however, putting them into an *Assemblies* will maintain there relative locations as will combining parts with boolean operations or within a *BuildPart* context.

As a example of creating parts with joints and connecting them together, consider the following code where flanges are attached to the ends of a curved pipe:



```
import copy
from build123d import *
from bd_warehouse.flange import WeldNeckFlange
from bd_warehouse.pipe import PipeSection
from ocp_vscode import *

flange_inlet = WeldNeckFlange(nps="10", flange_class=300)
flange_outlet = copy.copy(flange_inlet)
```

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```
with BuildPart() as pipe_builder:
    # Create the pipe
    with BuildLine():
        path = TangentArc((0, 0, 0), (2 * FT, 0, 1 * FT), tangent=(1, 0, 0))
    with BuildSketch(Plane(origin=path @ 0, z_dir=path % 0)):
        PipeSection("10", material="stainless", identifier="40S")
    sweep()

    # Add the joints
    RigidJoint(label="inlet", joint_location=-path.location_at(0))
    RigidJoint(label="outlet", joint_location=path.location_at(1))

# Place the flanges at the ends of the pipe
pipe_builder.part.joints["inlet"].connect_to(flange_inlet.joints["pipe"])
pipe_builder.part.joints["outlet"].connect_to(flange_outlet.joints["pipe"])

show(pipe_builder, flange_inlet, flange_outlet, render_joints=True)
```

Note how the locations of the joints are determined by the *location_at()* method and how the - negate operator is used to reverse the direction of the location without changing its position. Also note that the `WeldNeckFlange` class predefines two joints, one at the pipe end and one at the face end - both of which are shown in the above image (generated by ocp-vscode with the `render_joints=True` flag set in the `show` function).

class RigidJoint(*label: str*, *to_part: Solid | Compound | None = None*, *joint_location: Location | None = None*)

A rigid joint fixes two components to one another.

Parameters

- **label** (*str*) – joint label
- **to_part** (*Union[Solid, Compound], optional*) – object to attach joint to
- **joint_location** (Location) – global location of joint

Variables

relative_location (Location) – joint location relative to bound object

connect_to(*other:* BallJoint, *, *angles: Rotation | tuple[float, float, float] | None = None*, ***kwargs*)

connect_to(*other:* CylindricalJoint, *, *position: float | None = None*, *angle: float | None = None*)

connect_to(*other:* LinearJoint, *, *position: float | None = None*)

connect_to(*other:* RevoluteJoint, *, *angle: float | None = None*)

connect_to(*other:* RigidJoint)

Connect the RigidJoint to another Joint

Parameters

- **other** (Joint) – joint to connect to
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.
- **angles** (*RotationLike, optional*) – angles about axes in degrees. Defaults to range minimums.
- **position** (*float, optional*) – linear position. Defaults to linear range min.

property location: Location

Location of joint

relative_to(*other:* BallJoint, *, *angles: Rotation | tuple[float, float, float] | None = None*)

relative_to(*other:* CylindricalJoint, *, *position: float | None = None*, *angle: float | None = None*)

relative_to(*other:* LinearJoint, *, *position: float | None = None*)

relative_to(*other:* RevoluteJoint, *, *angle: float | None = None*)

relative_to(*other:* RigidJoint)

Relative location of RigidJoint to another Joint

Parameters

- **other** (RigidJoint) – relative to joint
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.
- **angles** (*RotationLike, optional*) – angles about axes in degrees. Defaults to range minimums.
- **position** (*float, optional*) – linear position. Defaults to linear range min.

Raises

TypeError – other must be of a type in: BallJoint, CylindricalJoint, LinearJoint, RevoluteJoint, RigidJoint.

property symbol: Compound

A CAD symbol (XYZ indicator) as bound to part

1.14.2 Revolute Joint

Component rotates around axis like a hinge. The *Joint Tutorial* covers Revolute Joints in detail.

During instantiation of a *RevoluteJoint* there are three parameters not present with Rigid Joints: `axis`, `angle_reference`, and `range` that allow the circular motion to be fully defined.

When *connect_to()* with a Revolute Joint, an extra `angle` parameter is present which allows one to change the relative position of joined parts by changing a single value.

class RevoluteJoint(*label: str*, *to_part: Solid | Compound | None = None*, *axis: Axis = Axis((0, 0, 0), (0, 0, 1))*, *angle_reference: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *angular_range: tuple[float, float] = (0, 360)*)

Component rotates around axis like a hinge.

Parameters

- **label** (*str*) – joint label
- **to_part** (*Union[Solid, Compound], optional*) – object to attach joint to
- **axis** (Axis) – axis of rotation
- **angle_reference** (*VectorLike, optional*) – direction normal to axis defining where angles will be measured from. Defaults to None.
- **range** (*tuple[float, float], optional*) – (min,max) angle of joint. Defaults to (0, 360).

Variables

- **angle** (*float*) – angle of joint
- **angle_reference** (Vector) – reference for angular positions

- **angular_range** (*tuple[float,float]*) – min and max angular position of joint
- **relative_axis** (Axis) – joint axis relative to bound part

Raises
ValueError – angle_reference must be normal to axis

connect_to(*other:* RigidJoint, *, *angle: float | None = None*)

Connect RevoluteJoint and RigidJoint

Parameters

- **other** (RigidJoint) – relative to joint
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.

Returns
other must of type RigidJoint ValueError: angle out of range

Return type
TypeError

property location: Location

Location of joint

relative_to(*other:* RigidJoint, *, *angle: float | None = None*)

Relative location of RevoluteJoint to RigidJoint

Parameters

- **other** (RigidJoint) – relative to joint
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.

Raises

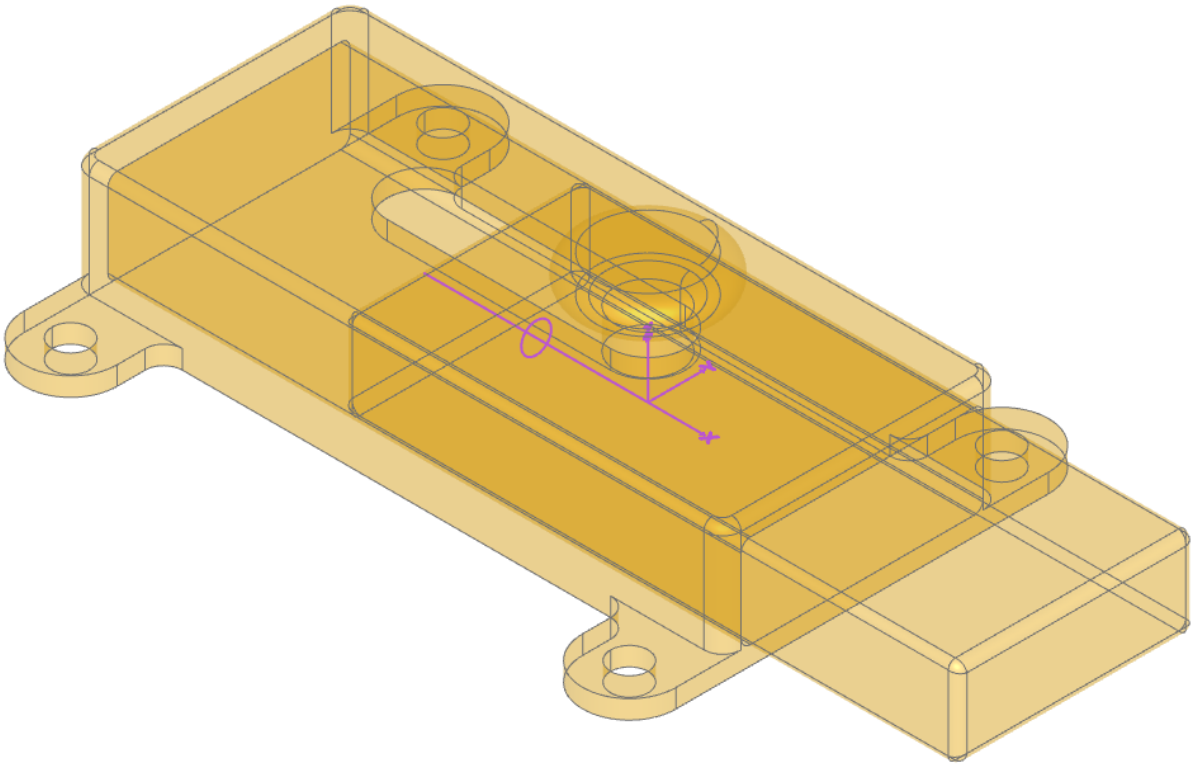
- **TypeError** – other must of type RigidJoint
- **ValueError** – angle out of range

property symbol: Compound

A CAD symbol representing the axis of rotation as bound to part

1.14.3 Linear Joint

Component moves along a single axis as with a sliding latch shown here:



The code to generate these components follows:

```
from build123d import *
from ocp_vscode import *

with BuildPart() as latch:
    # Basic box shape to start with filleted corners
    Box(70, 30, 14)
    end = latch.faces().sort_by(Axis.X)[-1]  # save the end with the hole
    fillet(latch.edges().filter_by(Axis.Z), 2)
    fillet(latch.edges().sort_by(Axis.Z)[-1], 1)
    # Make screw tabs
    with BuildSketch(latch.faces().sort_by(Axis.Z)[0]) as l4:
        with Locations((-30, 0), (30, 0)):
            SlotOverall(50, 10, rotation=90)
        Rectangle(50, 30)
        fillet(l4.vertices(Select.LAST), radius=2)
    extrude(amount=-2)
    with GridLocations(60, 40, 2, 2):
        Hole(2)
    # Create the hole from the end saved previously
    with BuildSketch(end) as slide_hole:
        add(end)
        offset(amount=-2)
```

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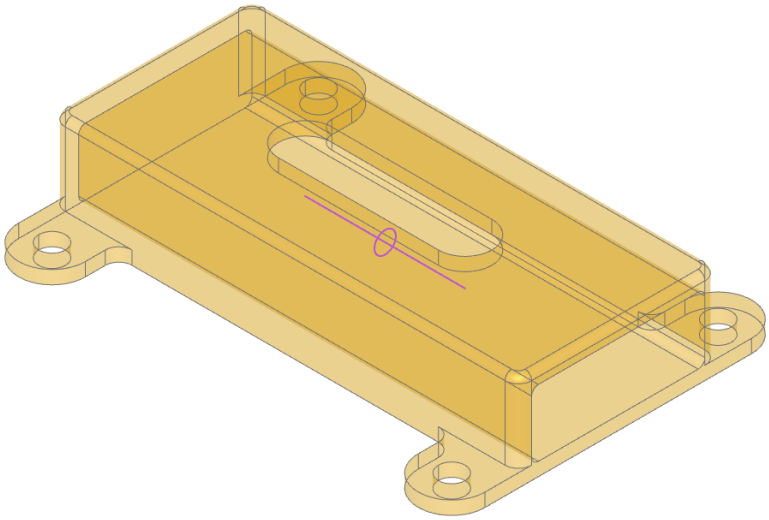
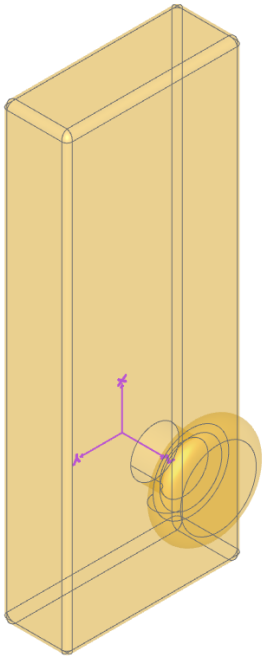
```
        fillet(slide_hole.vertices(), 1)
    extrude(amount=-68, mode=Mode.SUBTRACT)
    # Slot for the handle to slide in
    with BuildSketch(latch.faces().sort_by(Axis.Z)[-1]):
        SlotOverall(32, 8)
    extrude(amount=-2, mode=Mode.SUBTRACT)
    # The slider will move align the x axis 12mm in each direction
    LinearJoint("latch", axis=Axis.X, linear_range=(-12, 12))

with BuildPart() as slide:
    # The slide will be a little smaller than the hole
    with BuildSketch() as s1:
        add(slide_hole.sketch)
        offset(amount=-0.25)
    # The extrusions aren't symmetric
    extrude(amount=46)
    extrude(slide.faces().sort_by(Axis.Z)[0], amount=20)
    # Round off the ends
    fillet(slide.edges().group_by(Axis.Z)[0], 1)
    fillet(slide.edges().group_by(Axis.Z)[-1], 1)
    # Create the knob
    with BuildSketch() as s2:
        with Locations((12, 0)):
            SlotOverall(15, 4, rotation=90)
        Rectangle(12, 7, align=(Align.MIN, Align.CENTER))
        fillet(s2.vertices(Select.LAST), 1)
        split(bisect_by=Plane.XZ)
    revolve(axis=Axis.X)
    # Align the joint to Plane.ZY flipped
    RigidJoint("slide", joint_location=Location(-Plane.ZY))

# Position the slide in the latch: -12 >= position <= 12
latch.part.joints["latch"].connect_to(slide.part.joints["slide"], position=12)

# show(latch.part, render_joints=True)
# show(slide.part, render_joints=True)
show(latch.part, slide.part, render_joints=True)
```





Note how the slide is constructed in a different orientation than the direction of motion. The three highlighted lines of code show how the joints are created and connected together:

- The `LinearJoint` has an axis and limits of movement
- The `RigidJoint` has a single location, orientated such that the knob will ultimately be "up"
- The `connect_to` specifies a position that must be within the predefined limits.

The slider can be moved back and forth by just changing the `position` value. Values outside of the limits will raise an exception.

class LinearJoint(*label: str*, *to_part: Solid | Compound | None = None*, *axis: Axis = Axis((0, 0, 0), (0, 0, 1))*, *linear_range: tuple[float, float] = (0, inf)*)

Component moves along a single axis.

Parameters

- **label** (*str*) – joint label
- **to_part** (*Union[Solid, Compound], optional*) – object to attach joint to
- **axis** (Axis) – axis of linear motion
- **range** (*tuple[float, float], optional*) – (min,max) position of joint. Defaults to (0, inf).

Variables

- **axis** (Axis) – joint axis
- **angle** (*float*) – angle of joint
- **linear_range** (*tuple[float,float]*) – min and max positional values
- **position** (*float*) – joint position
- **relative_axis** (Axis) – joint axis relative to bound part

connect_to(*other:* RevoluteJoint, *, *position: float | None = None*, *angle: float | None = None*)

connect_to(*other:* RigidJoint, *, *position: float | None = None*)

Connect LinearJoint to another Joint

Parameters

- **other** (Joint) – joint to connect to
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.
- **position** (*float, optional*) – linear position. Defaults to linear range min.

Raises

- **TypeError** – other must be of type RevoluteJoint or RigidJoint
- **ValueError** – position out of range
- **ValueError** – angle out of range

property location: Location

Location of joint

relative_to(*other:* RigidJoint, *, *position: float | None = None*)

relative_to(*other:* RevoluteJoint, *, *position: float | None = None*, *angle: float | None = None*)

Relative location of LinearJoint to RevoluteJoint or RigidJoint

Parameters

- **other** (Joint) – joint to connect to
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.
- **position** (*float, optional*) – linear position. Defaults to linear range min.

Raises

- **TypeError** – other must be of type RevoluteJoint or RigidJoint
- **ValueError** – position out of range
- **ValueError** – angle out of range

property symbol: Compound

A CAD symbol of the linear axis positioned relative to_part

1.14.4 Cylindrical Joint

A *CylindricalJoint* allows a component to rotate around and moves along a single axis like a screw combining the functionality of a *LinearJoint* and a *RevoluteJoint* joint. The `connect_to` for these joints have both `position` and `angle` parameters as shown below extracted from the joint tutorial.

class CylindricalJoint(*label: str*, *to_part: Solid | Compound | None = None*, *axis: Axis = Axis((0, 0, 0), (0, 0, 1))*, *angle_reference: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *linear_range: tuple[float, float] = (0, inf)*, *angular_range: tuple[float, float] = (0, 360)*)

Component rotates around and moves along a single axis like a screw.

Parameters

- **label** (*str*) – joint label
- **to_part** (*Union[Solid, Compound], optional*) – object to attach joint to
- **axis** (Axis) – axis of rotation and linear motion
- **angle_reference** (*VectorLike, optional*) – direction normal to axis defining where angles will be measured from. Defaults to None.
- **linear_range** (*tuple[float, float], optional*) – (min,max) position of joint. Defaults to (0, inf).
- **angular_range** (*tuple[float, float], optional*) – (min,max) angle of joint. Defaults to (0, 360).

Variables

- **axis** (Axis) – joint axis
- **linear_position** (*float*) – linear joint position
- **rotational_position** (*float*) – revolute joint angle in degrees
- **angle_reference** (Vector) – reference for angular positions
- **angular_range** (*tuple[float,float]*) – min and max angular position of joint
- **linear_range** (*tuple[float,float]*) – min and max positional values
- **relative_axis** (Axis) – joint axis relative to bound part
- **position** (*float*) – joint position
- **angle** (*float*) – angle of joint

Raises

ValueError – angle_reference must be normal to axis

connect_to(*other: RigidJoint*, ***, *position: float | None = None*, *angle: float | None = None*)

Connect CylindricalJoint and RigidJoint"

Parameters

- **other** (Joint) – joint to connect to
- **position** (*float, optional*) – linear position. Defaults to linear range min.
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.

Raises

- **TypeError** – other must be of type RigidJoint

- **ValueError** – position out of range
- **ValueError** – angle out of range

property location: Location

Location of joint

relative_to(*other:* RigidJoint, *, *position: float | None = None*, *angle: float | None = None*)

Relative location of CylindricalJoint to RigidJoint

Parameters

- **other** (*Joint*) – joint to connect to
- **position** (*float, optional*) – linear position. Defaults to linear range min.
- **angle** (*float, optional*) – angle in degrees. Defaults to range min.

Raises

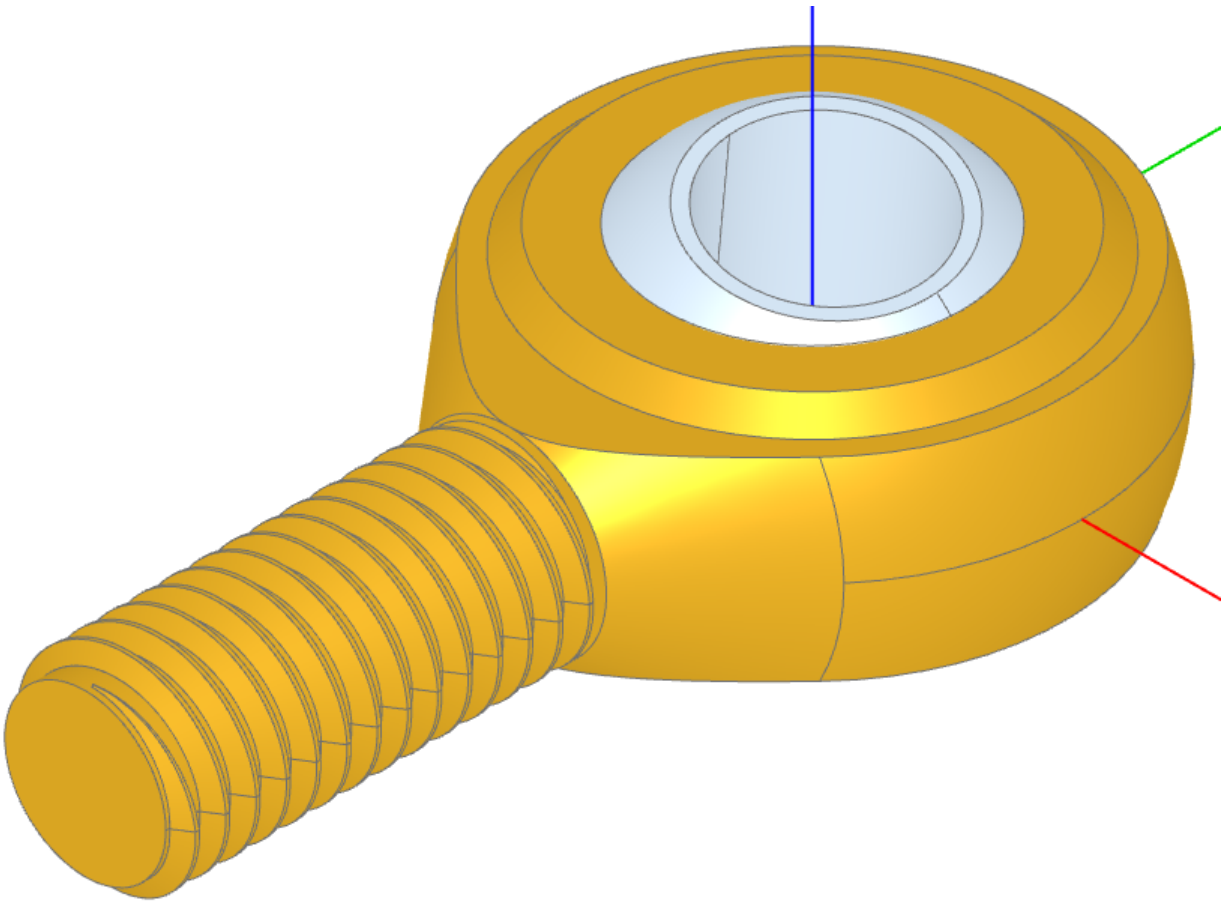
- **TypeError** – other must be of type RigidJoint
- **ValueError** – position out of range
- **ValueError** – angle out of range

property symbol: Compound

A CAD symbol representing the cylindrical axis as bound to part

1.14.5 Ball Joint

A component rotates around all 3 axes using a gimbal system (3 nested rotations). A *BallJoint* is found within a rod end as shown here:



```
from build123d import *
from bd_warehouse.thread import IsoThread
from ocp_vscode import *

# Create the thread so the min radius is available below
thread = IsoThread(major_diameter=6, pitch=1, length=20, end_finishes=("fade", "raw"))
inner_radius = 15.89 / 2
inner_gap = 0.2

with BuildPart() as rod_end:
    # Create the outer shape
    with BuildSketch():
        Circle(22.25 / 2)
        with Locations((0, -12)):
            Rectangle(8, 1)
        make_hull()
        split(bisect_by=Plane.YZ)
    revolve(axis=Axis.Y)
    # Refine the shape
    with BuildSketch(Plane.YZ) as s2:
        Rectangle(25, 8, align=(Align.MIN, Align.CENTER))
        Rectangle(9, 10, align=(Align.MIN, Align.CENTER))
        chamfer(s2.vertices(), 0.5)
```

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```
    revolve(axis=Axis.Z, mode=Mode.INTERSECT)
    # Add the screw shaft
    Cylinder(
        thread.min_radius,
        30,
        rotation=(90, 0, 0),
        align=(Align.CENTER, Align.CENTER, Align.MIN),
    )
    # Cutout the ball socket
    Sphere(inner_radius, mode=Mode.SUBTRACT)
    # Add thread
    with Locations((0, -30, 0)):
        add(thread, rotation=(-90, 0, 0))
    # Create the ball joint
    BallJoint(
        "socket",
        joint_location=Location(),
        angular_range=((-14, 14), (-14, 14), (0, 360)),
    )

with BuildPart() as ball:
    Sphere(inner_radius - inner_gap)
    Box(50, 50, 13, mode=Mode.INTERSECT)
    Hole(4)
    ball.part.color = Color("aliceblue")
    RigidJoint("ball", joint_location=Location())

rod_end.part.joints["socket"].connect_to(ball.part.joints["ball"], angles=(5, 10, 0))

show(rod_end.part, ball.part, s2)
```

Note how limits are defined during the instantiation of the ball joint when ensures that the pin or bolt within the rod end does not interfere with the rod end itself. The `connect_to` sets the three angles (only two are significant in this example).

class BallJoint(*label: str*, *to_part: Solid | Compound | None = None*, *joint_location: Location | None = None*, *angular_range: tuple[tuple[float, float], tuple[float, float], tuple[float, float]] = ((0, 360), (0, 360), (0, 360))*, *angle_reference: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*)

A component rotates around all 3 axes using a gimbal system (3 nested rotations).

Parameters

- **label** (*str*) – joint label
- **to_part** (*Union[Solid, Compound], optional*) – object to attach joint to
- **joint_location** (Location) – global location of joint
- **angular_range** – (tuple[tuple[float, float], tuple[float, float], tuple[float, float]], optional): X, Y, Z angle (min, max) pairs. Defaults to ((0, 360), (0, 360), (0, 360)).
- **angle_reference** (Plane, *optional*) – plane relative to part defining zero degrees of rotation. Defaults to Plane.XY.

Variables

- **relative_location** (Location) – joint location relative to bound part

- **angular_range** – (tuple[tuple[float, float], tuple[float, float], tuple[float, float]]): X, Y, Z angle (min, max) pairs.
- **angle_reference** (Plane) – plane relative to part defining zero degrees of

connect_to(*other:* RigidJoint, *, *angles: Rotation | tuple[float, float, float] | None = None*)

Connect BallJoint and RigidJoint

Parameters

- **other** (RigidJoint) – joint to connect to
- **angles** (*RotationLike, optional*) – angles about axes in degrees. Defaults to range minimums.

Raises

- **TypeError** – invalid other joint type
- **ValueError** – angles out of range

property location: Location

Location of joint

relative_to(*other:* RigidJoint, *, *angles: Rotation | tuple[float, float, float] | None = None*)

relative_to - BallJoint

Return the relative location from this joint to the RigidJoint of another object

Parameters

- **other** (RigidJoint) – joint to connect to
- **angles** (*RotationLike, optional*) – angles about axes in degrees. Defaults to range minimums.

Raises

- **TypeError** – invalid other joint type
- **ValueError** – angles out of range

property symbol: Compound

A CAD symbol representing joint as bound to part

1.15 Assemblies

Most CAD designs consist of more than one part which are naturally arranged in some type of assembly. Once parts have been assembled in a *Compound* object they can be treated as a unit - i.e. *moved()* or exported.

To create an assembly in build123d, one needs to create a tree of parts by simply assigning either a *Compound* object's `parent` or `children` attributes. To illustrate the process, we'll extend the *Joint Tutorial*.

1.15.1 Assigning Labels

In order keep track of objects one can assign a `label` to all *Shape* objects. Here we'll assign labels to all of the components that will be part of the box assembly:

```
box.label = "box"
lid.label = "lid"
hinge_outer.label = "outer hinge"
```

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```
hinge_inner.label = "inner hinge"
m6_screw.label = "M6 screw"
```

The labels are just strings with no further limitations (they don't have to be unique within the assembly).

1.15.2 Create the Assembly Compound

Creation of the assembly is done by simply creating a *Compound* object and assigning appropriate `parent` and `children` attributes as shown here:

```
box_assembly = Compound(label="assembly", children=[box, lid, hinge_inner, hinge_outer])
```

To display the topology of an assembly *Compound*, the *show_topology()* method can be used as follows:

```
print(box_assembly.show_topology())
```

which results in:

```
assembly        Compound at 0x7fc8ee235760, Location(p=(0, 0, 0), o=(-0, 0, -0))
├── box         Compound at 0x7fc8ee2188b0, Location(p=(0, 0, 50), o=(-0, 0, -0))
├── lid         Compound at 0x7fc8ee228460, Location(p=(-26, 0, 181), o=(-180, 30, -0))
├── inner hinge Hinge    at 0x7fc9292c3f70, Location(p=(-119, 60, 122), o=(90, 0, -150))
└── outer hinge Hinge    at 0x7fc9292c3f40, Location(p=(-150, 60, 50), o=(90, 0, 90))
```

To add to an assembly *Compound* one can change either `children` or `parent` attributes.

```
m6_screw.parent = box_assembly
print(box_assembly.show_topology())
```

and now the screw is part of the assembly.

```
assembly        Compound at 0x7fc8ee235760, Location(p=(0, 0, 0), o=(-0, 0, -0))
├── box         Compound at 0x7fc8ee2188b0, Location(p=(0, 0, 50), o=(-0, 0, -0))
├── lid         Compound at 0x7fc8ee228460, Location(p=(-26, 0, 181), o=(-180, 30, -0))
├── inner hinge Hinge    at 0x7fc9292c3f70, Location(p=(-119, 60, 122), o=(90, 0, -150))
├── outer hinge Hinge    at 0x7fc9292c3f40, Location(p=(-150, 60, 50), o=(90, 0, 90))
└── M6 screw    Compound at 0x7fc8ee235310, Location(p=(-157, -40, 70), o=(-0, -90, -60))
```

1.15.3 Shallow vs. Deep Copies of Shapes

Build123d supports the standard python `copy` module which provides two different types of copy operations `copy.copy()` and `copy.deepcopy()`.

Build123d's implementation of `deepcopy()` for the *Shape* class (e.g. `Solid`, `Face`, etc.) does just that, creates a complete copy of the original all the way down to the CAD object. `deepcopy` is therefore suited to the case where the copy will be subsequently modified to become its own unique item.

However, when building an assembly a common use case is to include many instances of an object, each one identical but in a different location. This is where `copy.copy()` is very useful as it copies all of the *Shape* except for the actual CAD object which instead is a reference to the original (OpenCascade refers this as a `TShape`). As it's a reference any changes to the original will be seen in all of the shallow copies.

Consider this example where 100 screws are added to an assembly:

```
screw = import_step("M6-1x12-countersunk-screw.step")
locs = HexLocations(6, 10, 10).local_locations

screw_copies = [copy.deepcopy(screw).locate(loc) for loc in locs]
copy_assembly = Compound(children=screw_copies)
export_step(copy_assembly, "copy_assembly.step")
```

which takes about 5 seconds to run (on an older computer) and produces a file of size 51938 KB. However, if a shallow copy is used instead:

```
screw = import_step("M6-1x12-countersunk-screw.step")
locs = HexLocations(6, 10, 10).local_locations

screw_references = [copy.copy(screw).locate(loc) for loc in locs]
reference_assembly = Compound(children=screw_references)
export_step(reference_assembly, "reference_assembly.step")
```

this takes about ¼ second and produces a file of size 550 KB - just over 1% of the size of the `deepcopy()` version and only 12% larger than the screw's step file.

Using `copy.copy()` to create references to the original CAD object for assemblies can substantially reduce the time and resources used to create and store that assembly.

1.15.4 Shapes are Anytree Nodes

The build123d assembly constructs are built using the python anytree package by making the build123d *Shape* class a sub-class of anytree's `NodeMixin` class. Doing so adds the following attributes to *Shape*:

- `parent` - Parent Node. On set, the node is detached from any previous parent node and attached to the new node.
- `children` - Tuple of all child nodes.
- `path` - Path of this `Node`.
- `iter_path_reverse` - Iterate up the tree from the current node.
- `ancestors` - All parent nodes and their parent nodes.
- `descendants` - All child nodes and all their child nodes.
- `root` - Tree Root Node.
- `siblings` - Tuple of nodes with the same parent.
- `leaves` - Tuple of all leaf nodes.
- `is_leaf` - `Node` has no children (External Node).
- `is_root` - `Node` is tree root.
- `height` - Number of edges on the longest path to a leaf `Node`.
- `depth` - Number of edges to the root `Node`.

Note

Changing the `children` attribute

Any iterator can be assigned to the `children` attribute but subsequently the children are stored as immutable `tuple` objects. To add a child to an existing *Compound* object, the `children` attribute will have to be reassigned.

1.15.5 Iterating Over Compounds

As Compounds are containers for shapes, build123d can iterate over these as required. Complex nested assemblies (compounds within compounds) do not need to be looped over with recursive functions. In the example below, the variable total_volume holds the sum of all the volumes in each solid in an assembly. Compare this to assembly3_volume which only results in the volume of the top level part.

```
# [import]
from build123d import *
from ocp_vscode import *

# Each assembly has a box and the previous assembly.
assembly1 = Compound(label='Assembly1', children=[Box(1, 1, 1),])
assembly2 = Compound(label='Assembly2', children=[assembly1, Box(1, 1, 1)])
assembly3 = Compound(label='Assembly3', children=[assembly2, Box(1, 1, 1)])
total_volume = sum(part.volume for part in assembly3.solids()) # 3
assembly3_volume = assembly3.volume # 1
```

1.15.6 pack

The *pack.pack()* function arranges objects in a compact, non-overlapping layout within a square(ish) 2D area. It is designed to minimize the space between objects while ensuring that no two objects overlap.

pack(*objects: Collection[Shape]*, *padding: float*, *align_z: bool = False*) → Collection[Shape]

Pack objects in a squarish area in Plane.XY.

Parameters

- **objects** (*Collection[Shape]*) – objects to arrange
- **padding** (*float*) – space between objects
- **align_z** (*bool, optional*) – align shape bottoms to Plane.XY. Defaults to False.

Returns
rearranged objects

Return type
Collection[Shape]

Detailed Description

The `pack` function uses a bin-packing algorithm to efficiently place objects within a 2D plane, ensuring that there is no overlap and that the space between objects is minimized. This is particularly useful in scenarios where spatial efficiency is crucial, such as layout design and object arrangement in constrained spaces.

The function begins by calculating the bounding boxes for each object, including the specified padding. It then uses a helper function `_pack2d` to determine the optimal positions for each object within the 2D plane. The positions are then translated back to the original objects, ensuring that they are arranged without overlapping.

Usage Note

The `align_z` parameter is especially useful when creating print-plates for 3D printing. By aligning the bottoms of the shapes to the same XY plane, you ensure that the objects are perfectly positioned for slicing software, which will no longer need to perform this alignment for you. This can streamline the process and improve the accuracy of the print setup.

Example Usage

```
# [import]
from build123d import *
from ocp_vscode import *


# [initial space]
b1 = Box(100, 100, 100, align=(Align.CENTER, Align.CENTER, Align.MIN))
b2 = Box(54, 54, 54, align=(Align.CENTER, Align.CENTER, Align.MAX), mode=Mode.SUBTRACT)
b3 = Box(34, 34, 34, align=(Align.MIN, Align.MIN, Align.CENTER), mode=Mode.SUBTRACT)
b4 = Box(24, 24, 24, align=(Align.MAX, Align.MAX, Align.CENTER), mode=Mode.SUBTRACT)
```

```
# [pack 2D]

xy_pack = pack(
    [b1, b2, b3, b4],
    padding=5,
    align_z=False
)
```

```
# [Pack and align_z]

z_pack = pack(
    [b1, b2, b3, b4],
    padding=5,
    align_z=True
)
```

Tip

If you place the arranged objects into a Compound, you can easily determine their bounding box and check whether the objects fit on your print bed.

```
# [bounding box]
print(Compound(xy_pack).bounding_box())
# bbox: 0.0 <= x <= 159.0, 0.0 <= y <= 129.0, -54.0 <= z <= 100.0

print(Compound(z_pack).bounding_box())
# bbox: 0.0 <= x <= 159.0, 0.0 <= y <= 129.0, 0.0 <= z <= 100.0
```

1.16 Tips, Best Practices and FAQ

Although there are countless ways to create objects with build123d, experience has proven that certain techniques can assist designers in achieving their goals with the greatest efficiency. The following is a description of these techniques.

1.16.1 Can't Get There from Here

Unfortunately, it's a reality that not all parts described using build123d can be successfully constructed by the underlying CAD core. Designers may have to explore different design approaches to get the OpenCascade CAD core to successfully build the target object. For instance, if a multi-section *sweep()* operation fails, a *loft()* operation may be a viable alternative in certain situations. It's crucial to remember that CAD is a complex field and patience may be required to achieve the desired results.

1.16.2 2D before 3D

When creating complex 3D objects, it is generally best to start with 2D work before moving on to 3D. This is because 3D structures are much more intricate, and 3D operations can be slower and more prone to failure. For designers who come from a Constructive Solid Geometry (CSG) background, such as OpenSCAD, this approach may seem counterintuitive. On the other hand, designers from a GUI BREP CAD background, like Fusion 360 or SolidWorks, may find this approach more natural.

In practice, this means that 3D objects are often created by applying operations like *extrude()* or *revolve()* to 2D sketches, as shown below:

```
with BuildPart() as my_part:
    with BuildSketch() as part_profile:
        ...
    extrude(amount=some_distance)
    ...
```

With this structure `part_profile` may have many objects that are combined and modified by operations like *fillet()* before being extruded to a 3D shape.

1.16.3 Delay Chamfers and Fillets

Chamfers and fillets can add complexity to a design by transforming simple vertices or edges into arcs or non-planar faces. This can significantly increase the complexity of the design. To avoid unnecessary processing costs and potential errors caused by a needlessly complicated design, it's recommended to perform these operations towards the end of the object's design. This is especially true for 3D shapes, as it is sometimes necessary to fillet or chamfer in the 2D design phase. Luckily, these 2D fillets and chamfers are less likely to fail than their 3D counterparts.

1.16.4 Parameterize

One of the most powerful features of build123d is the ability to design fully parameterized parts. While it may be faster to use a GUI CAD package for the initial iteration of a part, subsequent iterations can prove frustratingly difficult. By using variables for critical dimensions and deriving other dimensions from these key variables, not only can a single part be created, but a whole set of parts can be readily available. When inevitable change requests arise, a simple parameter adjustment may be all that's required to make necessary modifications.

1.16.5 Use Shallow Copies

As discussed in the Assembly section, a *shallow copy* of parts that are repeated in your design can make a huge difference in performance and usability of your end design. Objects like fasteners, bearings, chain links, etc. could be duplicated tens or even hundreds of times otherwise. Use shallow copies where possible but keep in mind that if one instance of the object changes all will change.

1.16.6 Object Selection

When selecting features in a design it's sometimes easier to select an object from higher up in the topology first, then select the object from there. For example let's consider a plate with four chamfered holes like this:

When selecting edges to be chamfered one might first select the face that these edges belong to then select the edges as shown here:

```
from build123d import *

svg_opts = {"pixel_scale": 5, "show_axes": False, "show_hidden": True}

length, width, thickness = 80.0, 60.0, 10.0
hole_dia = 6.0

with BuildPart() as plate:
    Box(length, width, thickness)
    with GridLocations(length - 20, width - 20, 2, 2):
        Hole(radius=hole_dia / 2)
    top_face: Face = plate.faces().sort_by(Axis.Z)[-1]
    hole_edges = top_face.edges().filter_by(GeomType.CIRCLE)
    chamfer(hole_edges, length=1)
```

1.16.7 Build123d - CadQuery Integration

As both CadQuery and **build123d** use a common OpenCascade Python wrapper (OCP) it's possible to interchange objects both from CadQuery to build123d and vice-versa by transferring the `wrapped` objects as follows (first from CadQuery to build123d):

```
import build123d as b3d
b3d_solid = b3d.Solid.make_box(1,1,1)

... some cadquery stuff ...

b3d_solid.wrapped = cq_solid.wrapped
```

Secondly, from build123d to CadQuery as follows:

```
import build123d as b3d
import cadquery as cq

with b3d.BuildPart() as b123d_box:
    b3d.Box(1,2,3)

cq_solid = cq.Solid.makeBox(1,1,1)
cq_solid.wrapped = b123d_box.part.solid().wrapped
```

1.16.8 Self Intersection

Avoid creating objects that intersect themselves - even if at a single vertex - as these topologies will almost certainly be invalid (even if *is_valid()* reports a `True` value). An example of where this may arise is with the thread of a screw (or any helical shape) where after one complete revolution the part may contact itself. One is likely be more successful if the part is split into multiple sections - say 180° of a helix - which are then stored in an assembly.

1.16.9 Packing Objects on a Plane

When designing independent shapes it's common to place each at or near the global origin, which can make it tricky to visualize many shapes at once. *pack.pack()* will translate the *Shape*'s passed to it so that they don't overlap, with an optional padding/spacing. Here's the result of packing a bunch of overlapping boxes (left) using some padding (right):

By default, the original Z value of all objects packed using the *pack.pack()* function is preserved. If you want to align all objects so that they are "placed" on the zero Z coordinate, the *pack()* function has an *align_z* argument. When set to *True*, this will align all objects.

This can be useful, for example, when preparing print setups for 3D printing, giving you full control over this alignment so you don't have to leave it to the slicer.

1.16.10 Isn't `from build123d import *` bad practice?

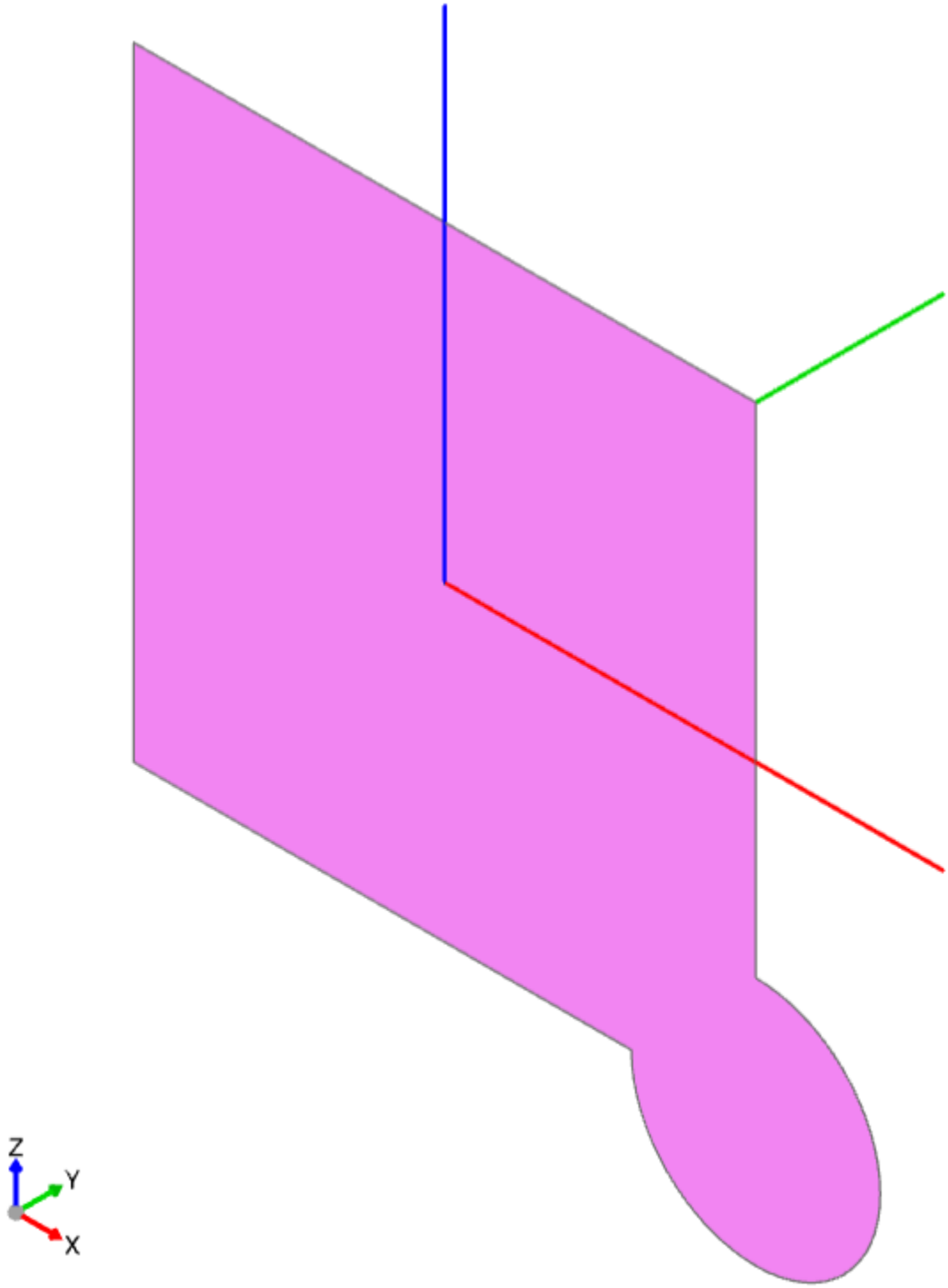
Glob imports like `from build123d import *` are generally frowned upon when writing software, and for good reason. They pollute the global namespace, cause confusing collisions, and are not future-proof, as future changes to the library being imported could collide with other names. It would be much safer to do something like `import build123d as bd` and then reference every item with, for example, `bd.BuildPart()`. If your goal is to integrate build123d into a larger piece of software, which many people work on, or where long-term maintainability is a priority, using this approach is definitely a good idea! Why then, are glob imports so often used in build123d code and official examples?

build123d is most commonly used not as a library within a larger application, but as a Domain-Specific Language which, together with something like the OCP CAD Viewer, acts as the user interface for a CAD application. Writing build123d often involves live coding in a REPL or typing in editors with limited space due to the rest of the CAD GUI taking up screen space. Scripts are usually centred around build123d usage, with usage of other libraries being limited enough that naming conflicts are easily avoided. In this context, it's entirely reasonable to prioritise developer ergonomics over "correctness" by making build123d's primitives available in the global namespace.

1.16.11 Why doesn't BuildSketch(Plane.XZ) work?

When creating a sketch not on the default `Plane.XY` users may expect that they are drawing directly on the workplane / coordinate system provided. For example:

```
with BuildSketch(Plane.XZ) as vertical_sketch:
    Rectangle(1, 1)
    with Locations(vertices().group_by(Axis.X)[-1].sort_by(Axis.Z)[-1]):
        Circle(0.2)
```

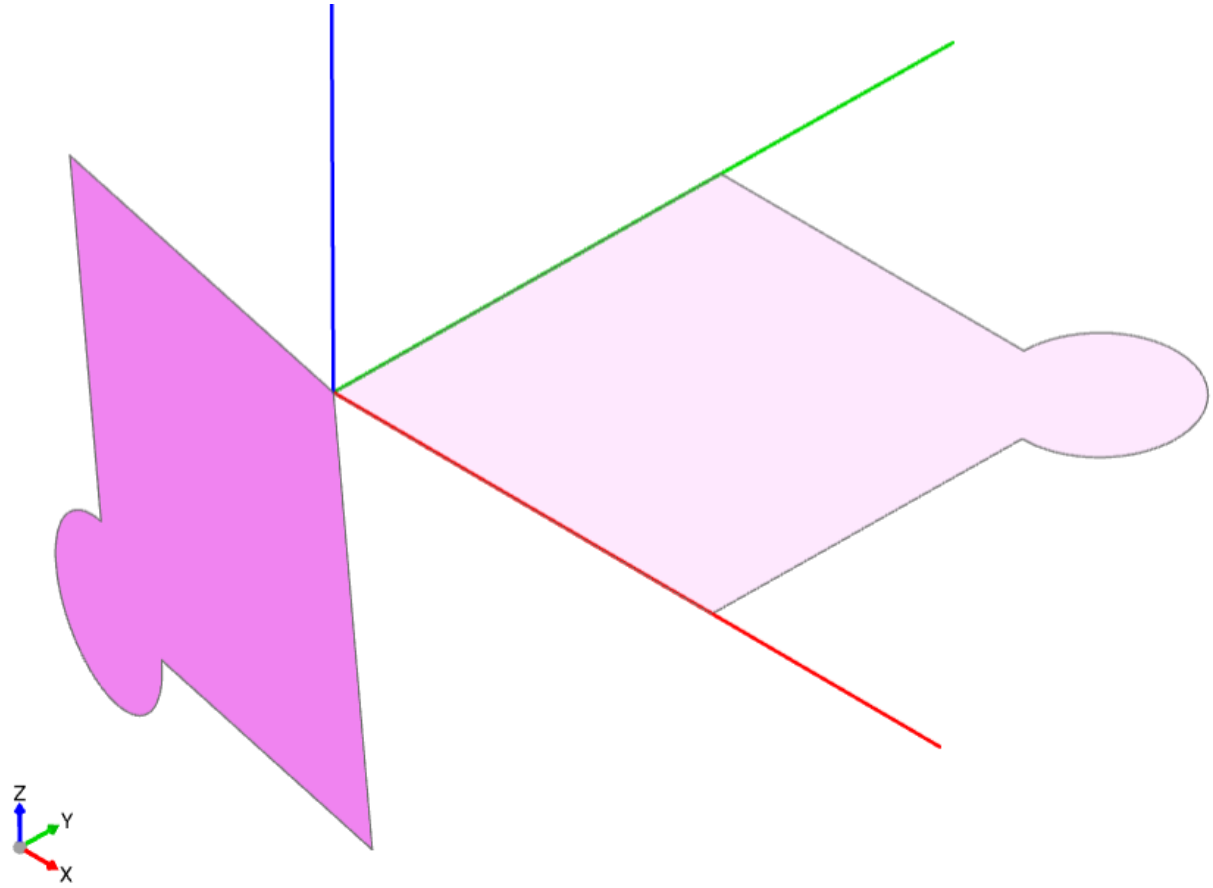


In this case the circle is not positioned in the top right as one would expect; in-fact, the position of the circle randomly switches between the bottom and top corner.

This is because all sketches are created on a local `Plane.XY` independent of where they will be ultimately placed; therefore, the `sort_by(Axis.Z)` is sorting two points that have a Z value of zero as they are located on `Plane.XY` and effectively return a random point.

Why does BuildSketch work this way? Consider an example where the user wants to work on a plane not aligned with any Axis, as follows (this is often done when creating a sketch on a Face of a 3D part but is simulated here by rotating a Plane):

```
with BuildSketch(Plane.YZ.rotated((123, 45, 6))) as custom_plane:
    Rectangle(1, 1, align=Align.MIN)
    with Locations(vertices().group_by(Axis.X)[-1].sort_by(Axis.Y)[-1]):
        Circle(0.2)
```



Here one can see both sketch_local (with the light fill on Plane.XY) and the sketch (with the darker fill) placed on the user provided workplane. As the selectors work off global coordinates, selection of the "top right" of this sketch would be quite challenging and would likely change if the sketch was ever moved as could happen if the 3D part changed. For an example of sketching on a 3D part, see *Sketching on other Planes*.

1.16.12 Why is BuildLine not working as expected within the scope of BuildSketch?

As described above, all sketching is done on a local Plane.XY; however, the following is a common issue:

```
with BuildSketch() as sketch:
    with BuildLine(Plane.XZ):
        Polyline(...)
    make_face()
```

Here BuildLine is within the scope of BuildSketch; therefore, all of the drawing should be done on Plane.XY; however, the user has specified Plane.XZ when creating the BuildLine instance. Although this isn't absolutely incorrect it's almost certainly not what the user intended. Here the face created by make_face will be reoriented to

`Plane.XY` as all sketching must be done on that plane. This reorienting of objects to `Plane.XY` allows a user to `add` content from other sources to the sketch without having to manually re-orient the object.

Unless there is a good reason and the user understands how the `BuildLine` object will be reoriented, all `BuildLine` instances within the scope of `BuildSketch` should be done on the default `Plane.XY`.

1.16.13 Don't Builders inherit workplane/coordinate systems when nested

Some users expect that nested Builders will inherit the workplane or coordinate system from their parent Builder - this is not true. When a Builder is instantiated, a workplane is either provided by the user or it defaults to `Plane.XY`. Having Builders inherent coordinate systems from their parents could result in confusion when they are nested as well as change their behaviour depending on which scope they are in. Inheriting coordinate systems isn't necessarily incorrect, it was considered for build123d but ultimately the simple static approach was taken.

1.17 Import/Export

Methods and functions specific to exporting and importing build123d objects are defined below.

For example:

```
with BuildPart() as box_builder:
    Box(1, 1, 1)
export_step(box_builder.part, "box.step")
```

1.17.1 File Formats

3MF

The 3MF (3D Manufacturing Format) file format is a versatile and modern standard for representing 3D models used in additive manufacturing, 3D printing, and other applications. Developed by the 3MF Consortium, it aims to overcome the limitations of traditional 3D file formats by providing a more efficient and feature-rich solution. The 3MF format supports various advanced features like color information, texture mapping, multi-material definitions, and precise geometry representation, enabling seamless communication between design software, 3D printers, and other manufacturing devices. Its open and extensible nature makes it an ideal choice for exchanging complex 3D data in a compact and interoperable manner.

BREP

The BREP (Boundary Representation) file format is a widely used data format in computer-aided design (CAD) and computer-aided engineering (CAE) applications. BREP represents 3D geometry using topological entities like vertices, edges, and faces, along with their connectivity information. It provides a precise and comprehensive representation of complex 3D models, making it suitable for advanced modeling and analysis tasks. BREP files are widely supported by various CAD software, enabling seamless data exchange between different systems. Its ability to represent both geometric shapes and their topological relationships makes it a fundamental format for storing and sharing detailed 3D models.

DXF

The DXF (Drawing Exchange Format) file format is a widely used standard for representing 2D and 3D drawings, primarily used in computer-aided design (CAD) applications. Developed by Autodesk, DXF files store graphical and geometric data, such as lines, arcs, circles, and text, as well as information about layers, colors, and line weights. Due to its popularity, DXF files can be easily exchanged and shared between different CAD software. The format's simplicity and human-readable structure make it a versatile choice for sharing designs, drawings, and models across various CAD platforms, facilitating seamless collaboration in engineering and architectural projects.

glTF

The glTF (GL Transmission Format) is a royalty-free specification for the efficient transmission and loading of 3D models and scenes by applications. Developed by the Khronos Group, glTF is designed as a compact, interoperable format that enables the quick display of assets across various platforms and devices. glTF supports a rich feature set, including detailed meshes, materials, textures, skeletal animations, and more, facilitating complex 3D visualizations. It streamlines the process of sharing and deploying 3D content in web applications, game engines, and other visualization tools, making it the "JPEG of 3D." glTF's versatility and efficiency have led to its widespread adoption in the 3D content industry.

STL

The STL (STereoLithography) file format is a widely used file format in 3D printing and computer-aided design (CAD) applications. It represents 3D geometry using triangular facets to approximate the surface of a 3D model. STL files are widely supported and can store both the geometry and color information of the model. They are used for rapid prototyping and 3D printing, as they provide a simple and efficient way to represent complex 3D objects. The format's popularity stems from its ease of use, platform independence, and ability to accurately describe the surface of intricate 3D models with a minimal file size.

STEP

The STEP (Standard for the Exchange of Product model data) file format is a widely used standard for representing 3D product and manufacturing data in computer-aided design (CAD) and computer-aided engineering (CAE) applications. It is an ISO standard (ISO 10303) and supports the representation of complex 3D geometry, product structure, and metadata. STEP files store information in a neutral and standardized format, making them highly interoperable across different CAD/CAM software systems. They enable seamless data exchange between various engineering disciplines, facilitating collaboration and data integration throughout the entire product development and manufacturing process.

SVG

The SVG (Scalable Vector Graphics) file format is an XML-based standard used for describing 2D vector graphics. It is widely supported and can be displayed in modern web browsers, making it suitable for web-based graphics and interactive applications. SVG files define shapes, paths, text, and images using mathematical equations, allowing for smooth scalability without loss of quality. The format is ideal for logos, icons, illustrations, and other graphics that require resolution independence. SVG files are also easily editable in text editors or vector graphic software, making them a popular choice for designers and developers seeking flexible and versatile graphic representation.

1.17.2 2D Exporters

Exports to DXF (Drawing Exchange Format) and SVG (Scalable Vector Graphics) are provided by the 2D Exporters: ExportDXF and ExportSVG classes.

DXF is a widely used file format for exchanging CAD (Computer-Aided Design) data between different software applications. SVG is a widely used vector graphics format that is supported by web browsers and various graphic editors.

The core concept to these classes is the creation of a DXF/SVG document with specific properties followed by the addition of layers and shapes to the documents. Once all of the layers and shapes have been added, the document can be written to a file.

3D to 2D Projection

There are a couple ways to generate a 2D drawing of a 3D part:

- Generate a section: The *section()* operation can be used to create a 2D cross section of a 3D part at a given plane.
- Generate a projection: The `project_to_viewport()` method can be used to create a 2D projection of a 3D scene. Similar to a camera, the `viewport_origin` defines the location of camera, the `viewport_up` defines

the orientation of the camera, and the `look_at` parameter defined where the camera is pointed. By default, `viewport_up` is the positive z axis and `look_up` is the center of the shape. The return value is a tuple of lists of edges, the first the visible edges and the second the hidden edges.

Each of these Edges and Faces can be assigned different line color/types and fill colors as described below (as `project_to_viewport` only generates Edges, fill doesn't apply). The shapes generated from the above steps are to be added as shapes in one of the exporters described below and written as either a DXF or SVG file as shown in this example:

```
view_port_origin=(-100, -50, 30)
visible, hidden = part.project_to_viewport(view_port_origin)
max_dimension = max(*Compound(children=visible + hidden).bounding_box().size)
exporter = ExportSVG(scale=100 / max_dimension)
exporter.add_layer("Visible")
exporter.add_layer("Hidden", line_color=(99, 99, 99), line_type=LineType.ISO_DOT)
exporter.add_shape(visible, layer="Visible")
exporter.add_shape(hidden, layer="Hidden")
exporter.write("part_projection.svg")
```

LineType

ANSI (American National Standards Institute) and ISO (International Organization for Standardization) standards both define line types in drawings used in DXF and SVG exported drawings:

- **ANSI Standards:**
  - ANSI/ASME Y14.2 - "Line Conventions and Lettering" is the standard that defines line types, line weights, and line usage in engineering drawings in the United States.
- **ISO Standards:**
  - ISO 128 - "Technical drawings – General principles of presentation" is the ISO standard that covers the general principles of technical drawing presentation, including line types and line conventions.
  - ISO 13567 - "Technical product documentation (TPD) – Organization and naming of layers for CAD" provides guidelines for the organization and naming of layers in Computer-Aided Design (CAD) systems, which may include line type information.

These standards help ensure consistency and clarity in technical drawings, making it easier for engineers, designers, and manufacturers to communicate and interpret the information presented in the drawings.

The line types used by the 2D Exporters are defined by the `LineType` Enum and are shown in the following diagram:

ExportDXF

class ExportDXF(*version: str = 'AC1027'*, *unit: ~build123d.build_enums.Unit = <Unit.MM>*, *color: ~exporters.ColorIndex | None = None*, *line_weight: float | None = None*, *line_type: ~exporters.LineType | None = None*)

The ExportDXF class provides functionality for exporting 2D shapes to DXF (Drawing Exchange Format) format. DXF is a widely used file format for exchanging CAD (Computer-Aided Design) data between different software applications.

Parameters

- **version** (*str, optional*) – The DXF version to use for the output file. Defaults to ezdxf.DXF2013.

- **unit** (*Unit, optional*) – The unit used for the exported DXF. It should be one of the Unit enums: Unit.MC, Unit.MM, Unit.CM, Unit.M, Unit.IN, or Unit.FT. Defaults to Unit.MM.
- **color** (*Optional[ColorIndex], optional*) – The default color index for shapes. It can be specified as a ColorIndex enum or None.. Defaults to None.
- **line_weight** (*Optional[float], optional*) – The default line weight (stroke width) for shapes, in millimeters. . Defaults to None.
- **line_type** (*Optional[LineType], optional*) – e default line type for shapes. It should be a LineType enum or None.. Defaults to None.

Example

```
exporter = ExportDXF(unit=Unit.MM, line_weight=0.5)
exporter.add_layer("Layer 1", color=ColorIndex.RED, line_type=LineType.DASHED)
exporter.add_shape(shape_object, layer="Layer 1")
exporter.write("output.dxf")
```

Raises
ValueError – unit not supported

METRIC_UNITS = {<Unit.CM>, <Unit.M>, <Unit.MM>}

add_layer(*name: str*, *, *color: ColorIndex | None = None*, *line_weight: float | None = None*, *line_type: LineType | None = None*) → Self

Adds a new layer to the DXF export with the given properties.

Parameters

- **name** (*str*) – The name of the layer definition. Must be unique among all layers.
- **color** (*Optional[ColorIndex], optional*) – The color index for shapes on this layer. It can be specified as a ColorIndex enum or None. Defaults to None.
- **line_weight** (*Optional[float], optional*) – The line weight (stroke width) for shapes on this layer, in millimeters. Defaults to None.
- **line_type** (*Optional[LineType], optional*) – The line type for shapes on this layer. It should be a LineType enum or None. Defaults to None.

Returns
DXF document with additional layer

Return type
Self

add_shape(*shape: Shape | Iterable[Shape]*, *layer: str = ''*) → Self

Adds a shape to the specified layer.

Parameters

- **shape** (*Shape | Iterable[Shape]*) – The shape or collection of shapes to be added. It can be a single Shape object or an iterable of Shape objects.
- **layer** (*str, optional*) – The name of the layer where the shape will be added. If not specified, the default layer will be used. Defaults to “”.

Returns
Document with additional shape

Return type
Self

write(*file_name: PathLike | str | bytes | BytesIO*, *ascii_format: bool = True*)

Writes the DXF data to the specified file name.

Parameters

- **file_name** (*PathLike | str | bytes | BytesIO*) – The file name (including path) where the DXF data will be written.
- **ascii_format** (*bool, optional*) – Export the file as ASCII (True) or binary (False) DXF format. Defaults to True.

ExportSVG

class ExportSVG(*unit: ~build123d.build_enums.Unit = <Unit.MM>*, *scale: float = 1*, *margin: float = 0*, *fit_to_stroke: bool = True*, *precision: int = 6*, *fill_color: ~exporters.ColorIndex | ~ezdxf.colors.RGB | ~build123d.geometry.Color | None = None*, *line_color: ~exporters.ColorIndex | ~ezdxf.colors.RGB | ~build123d.geometry.Color | None = ColorIndex.BLACK*, *line_weight: float = 0.09*, *line_type: ~exporters.LineType = LineType.CONTINUOUS*, *dot_length: ~exporters.DotLength | float = DotLength.INKSCAPE_COMPAT*)

SVG file export functionality.

The ExportSVG class provides functionality for exporting 2D shapes to SVG (Scalable Vector Graphics) format. SVG is a widely used vector graphics format that is supported by web browsers and various graphic editors.

Parameters

- **unit** (*Unit, optional*) – The unit used for the exported SVG. It should be one of the Unit enums: Unit.MM, Unit.CM, or Unit.IN. Defaults to Unit.MM.
- **scale** (*float, optional*) – The scaling factor applied to the exported SVG. Defaults to 1.
- **margin** (*float, optional*) – The margin added around the exported shapes. Defaults to 0.
- **fit_to_stroke** (*bool, optional*) – A boolean indicating whether the SVG view box should fit the strokes of the shapes. Defaults to True.
- **precision** (*int, optional*) – The number of decimal places used for rounding coordinates in the SVG. Defaults to 6.
- **fill_color** (*ColorIndex | RGB | None, optional*) – The default fill color for shapes. It can be specified as a ColorIndex, an RGB tuple, or None. Defaults to None.
- **line_color** (*ColorIndex | RGB | None, optional*) – The default line color for shapes. It can be specified as a ColorIndex or an RGB tuple, or None. Defaults to Export2D.DEFAULT_COLOR_INDEX.
- **line_weight** (*float, optional*) – The default line weight (stroke width) for shapes, in millimeters. Defaults to Export2D.DEFAULT_LINE_WEIGHT.
- **line_type** (*LineType, optional*) – The default line type for shapes. It should be a LineType enum. Defaults to Export2D.DEFAULT_LINE_TYPE.
- **dot_length** (*DotLength | float, optional*) – The width of rendered dots in a Can be either a DotLength enum or a float value in tenths of an inch. Defaults to DotLength.INKSCAPE_COMPAT.

Example

```
exporter = ExportSVG(unit=Unit.MM, line_weight=0.5)
exporter.add_layer("Layer 1", fill_color=(255, 0, 0), line_color=(0, 0, 255))
exporter.add_shape(shape_object, layer="Layer 1")
exporter.write("output.svg")
```

Raises
ValueError – Invalid unit.

add_layer(*name: str*, ***, *fill_color: ColorIndex | RGB | Color | None = None*, *line_color: ColorIndex | RGB | Color | None = ColorIndex.BLACK*, *line_weight: float = 0.09*, *line_type: LineType = LineType.CONTINUOUS*) → Self

Adds a new layer to the SVG export with the given properties.

Parameters

- **name** (*str*) – The name of the layer. Must be unique among all layers.
- **fill_color** (*ColorIndex | RGB | Color | None, optional*) – The fill color for shapes on this layer. It can be specified as a ColorIndex, an RGB tuple, a Color, or None. Defaults to None.
- **line_color** (*ColorIndex | RGB | Color | None, optional*) – The line color for shapes on this layer. It can be specified as a ColorIndex or an RGB tuple, a Color, or None. Defaults to Export2D.DEFAULT_COLOR_INDEX.
- **line_weight** (*float, optional*) – The line weight (stroke width) for shapes on this layer, in millimeters. Defaults to Export2D.DEFAULT_LINE_WEIGHT.
- **line_type** (*LineType, optional*) – The line type for shapes on this layer. It should be a LineType enum. Defaults to Export2D.DEFAULT_LINE_TYPE.

Raises

- **ValueError** – Duplicate layer name
- **ValueError** – Unknown linetype

Returns
Drawing with an additional layer

Return type
Self

add_shape(*shape: Shape | Iterable[Shape]*, *layer: str = ''*, *reverse_wires: bool = False*)

Adds a shape or a collection of shapes to the specified layer.

Parameters

- **shape** (*Shape | Iterable[Shape]*) – The shape or collection of shapes to be added. It can be a single Shape object or an iterable of Shape objects.
- **layer** (*str, optional*) – The name of the layer where the shape(s) will be added. Defaults to “”.
- **reverse_wires** (*bool, optional*) – A boolean indicating whether the wires of the shape(s) should be in reversed direction. Defaults to False.

Raises
ValueError – Undefined layer

write(*path: PathLike | str | bytes | BytesIO*)

Writes the SVG data to the specified file path.

Parameters

path (*PathLike | str | bytes | BytesIO*) – The file path where the SVG data will be written.

1.17.3 3D Exporters

export_brep(*to_export: Shape*, *file_path: PathLike | str | bytes | BytesIO | BinaryIO*) → bool

Export this shape to a BREP file

Parameters

- **to_export** (Shape) – object or assembly
- **file_path** – Union[PathLike, str, bytes, BytesIO]: brep file path or memory buffer

Returns

write status

Return type

bool

export_gltf(*to_export: ~build123d.topology.shape_core.Shape*, *file_path: ~os.PathLike | str | bytes*, *unit: ~build123d.build_enums.Unit = <Unit.MM>*, *binary: bool = False*, *linear_deflection: float = 0.001*, *angular_deflection: float = 0.1*) → bool

The glTF (GL Transmission Format) specification primarily focuses on the efficient transmission and loading of 3D models as a compact, binary format that is directly renderable by graphics APIs like WebGL, OpenGL, and Vulkan. It's designed to store detailed 3D model data, including meshes (vertices, normals, textures, etc.), animations, materials, and scene hierarchy, among other aspects.

Parameters

- **to_export** (Shape) – object or assembly
- **file_path** (*Union[PathLike, str, bytes]*) – glTF file path
- **unit** (*Unit, optional*) – shape units. Defaults to Unit.MM.
- **binary** (*bool, optional*) – output format. Defaults to False.
- **linear_deflection** (*float, optional*) – A linear deflection setting which limits the distance between a curve and its tessellation. Setting this value too low will result in large meshes that can consume computing resources. Setting the value too high can result in meshes with a level of detail that is too low. The default is a good starting point for a range of cases. Defaults to 1e-3.
- **angular_deflection** (*float, optional*) – Angular deflection setting which limits the angle between subsequent segments in a polyline. Defaults to 0.1.

Raises

RuntimeError – Failed to write glTF file

Returns

write status

Return type

bool

export_step(*to_export: ~build123d.topology.shape_core.Shape*, *file_path: ~os.PathLike | str | bytes | ~_io.BytesIO | ~typing.BinaryIO*, *unit: ~build123d.build_enums.Unit = <Unit.MM>*, *write_pcurves: bool = True*, *precision_mode: ~build123d.build_enums.PrecisionMode = <PrecisionMode.AVERAGE>*, ***, *timestamp: str | ~datetime.datetime | None = None*) → bool

Export a build123d Shape or assembly with color and label attributes. Note that if the color of a node in an assembly isn't set, it will be assigned the color of its nearest ancestor.

Parameters

- **to_export** (Shape) – object or assembly
- **file_path** (*Union[PathLike, str, bytes, BytesIO]*) – step file path
- **unit** (*Unit, optional*) – shape units. Defaults to Unit.MM.
- **write_pcurves** (*bool, optional*) – write parametric curves to the STEP file. Defaults to True.
- **precision_mode** (*PrecisionMode, optional*) – geometric data precision. Defaults to PrecisionMode.AVERAGE.

Raises
RuntimeError – Unknown Compound type

Returns
success

Return type
bool

export_stl(*to_export: Shape*, *file_path: PathLike | str | bytes*, *tolerance: float = 0.001*, *angular_tolerance: float = 0.1*, *ascii_format: bool = False*) → bool

Export STL

Exports a shape to a specified STL file.

Parameters

- **to_export** (Shape) – object or assembly
- **file_path** (*Union[PathLike, str, bytes]*) – The path and file name to write the STL output to.
- **tolerance** (*float, optional*) – A linear deflection setting which limits the distance between a curve and its tessellation. Setting this value too low will result in large meshes that can consume computing resources. Setting the value too high can result in meshes with a level of detail that is too low. The default is a good starting point for a range of cases. Defaults to 1e-3.
- **angular_tolerance** (*float, optional*) – Angular deflection setting which limits the angle between subsequent segments in a polyline. Defaults to 0.1.
- **ascii_format** (*bool, optional*) – Export the file as ASCII (True) or binary (False) STL format. Defaults to False (binary).

Returns
Success

Return type
bool

3D Mesh Export

Both 3MF and STL export (and import) are provided with the *Mesher* class. As mentioned above, the 3MF format it provides is feature-rich and therefore has a slightly more complex API than the simple Shape exporters.

For example:

```
# Create the shapes and assign attributes
blue_shape = Solid.make_cone(20, 0, 50)
blue_shape.color = Color("blue")
blue_shape.label = "blue"
blue_uuid = uuid.uuid1()
red_shape = Solid.make_cylinder(5, 50).move(Location((0, -30, 0)))
red_shape.color = Color("red")
red_shape.label = "red"

# Create a Mesher instance as an exporter, add shapes and write
exporter = Mesher()
exporter.add_shape(blue_shape, part_number="blue-1234-5", uuid_value=blue_uuid)
exporter.add_shape(red_shape)
exporter.add_meta_data(
    name_space="custom",
    name="test_meta_data",
    value="hello world",
    metadata_type="str",
    must_preserve=False,
)
exporter.add_code_to_metadata()
exporter.write("example.3mf")
exporter.write("example.stl")
```

class Mesher(*unit: ~build123d.build_enums.Unit = <Unit.MM>*)

Tool for exporting and importing meshed objects stored in 3MF or STL files.

Parameters
unit (*Unit, optional*) – model units. Defaults to Unit.MM.

add_code_to_metadata()

Add the code calling this method to the 3MF metadata with the custom name space *build123d*, name equal to the base file name and the type as *python*

add_meta_data(*name_space: str*, *name: str*, *value: str*, *metadata_type: str*, *must_preserve: bool*)

Add meta data to the models

Parameters

- **name_space** (*str*) – categorizer of different metadata entries
- **name** (*str*) – metadata label
- **value** (*str*) – metadata content
- **metadata_type** (*str*) – metadata type
- **must_preserve** (*bool*) – metadata must not be removed if unused

add_shape(*shape: ~build123d.topology.shape_core.Shape | ~collections.abc.Iterable[~build123d.topology.shape_core.Shape], linear_deflection: float = 0.001, angular_deflection: float = 0.1, mesh_type: ~build123d.build_enums.MeshType = <MeshType.MODEL>, part_number: str | None = None, uuid_value: ~uuid.UUID | None = None*)

Add a shape to the 3MF/STL file.

Parameters

- **shape** (*Union[Shape, Iterable[Shape]]*) – build123d object
- **linear_deflection** (*float, optional*) – mesh control for edges. Defaults to 0.001.
- **angular_deflection** (*float, optional*) – mesh control for non-planar surfaces. Defaults to 0.1.
- **mesh_type** (*MeshType, optional*) – 3D printing use of mesh. Defaults to MeshType.MODEL.
- **part_number** (*str, optional*) – part #. Defaults to None.
- **uuid_value** (*uuid, optional*) – value from uuid package. Defaults to None.

Raises

- **RuntimeError** – 3mf mesh is invalid
- **Warning** – Degenerate shape skipped
- **Warning** – 3mf mesh is not manifold

get_mesh_properties() → list[dict]

Retrieve the properties from all the meshes

get_meta_data() → list[dict]

Retrieve all of the metadata

get_meta_data_by_key(*name_space: str*, *name: str*) → dict

Retrieve the metadata value and type for the provided name space and name

property library_version: str

3MF Consortium Lib#MF version

property mesh_count: int

Number of meshes in the model

property model_unit: Unit

Unit used in the model

read(*file_name: PathLike | str | bytes*) → list[Shape]

Parameters

- **Union[PathLike** (*file_name*) – file path
- **str** – file path
- **bytes]** – file path

Raises

ValueError – Unknown file format - must be 3mf or stl

Returns

build123d shapes extracted from mesh file

Return type
list[Shape]

property triangle_counts: list[int]
Number of triangles in each of the model's meshes

property vertex_counts: list[int]
Number of vertices in each of the models's meshes

write(*file_name: PathLike | str | bytes*)

Parameters
- **Union[Pathlike** (*file_name*) – file path
- **str** – file path
- **bytes]** – file path

Raises
ValueError – Unknown file format - must be 3mf or stl

write_stream(*stream: BytesIO*, *file_type: Literal['3mf', 'stl']*)

Parameters
- **stream** (*BytesIO*) – byte stream
- **file_type** – output mesh format, either "3mf" or "stl"

Raises
ValueError – Unknown file format - must be 3mf or stl

Note

If you need to align multiple components for 3D printing, you can use the *pack()* function to arrange the objects side by side and align them on the same plane. This ensures that your components are well-organized and ready for the printing process.

1.17.4 2D Importers

import_dxf(*dxf_file: str | PathLike | TextIO | BinaryIO*) → ShapeList
Import shapes from a DXF file

Parameters
dxf_file (*str | PathLike | TextIO | BinaryIO*) – dxf file path or readable stream

Raises
DXFStructureError – file not found

Returns
build123d objects

Return type
ShapeList

import_svg(*svg_file: str | Path | TextIO*, ***, *flip_y: bool = True*, *align: Align | tuple[Align, Align] | None = Align.MIN*, *ignore_visibility: bool = False*, *label_by: Literal['id', 'class', 'inkscape:label'] | str = 'id'*) → ShapeList[Wire | Face]

import_svg(*svg_file: str | Path | TextIO*, ***, *flip_y: bool = True*, *align: Align | tuple[Align, Align] | None = Align.MIN*, *ignore_visibility: bool = False*, *label_by: Literal['id', 'class', 'inkscape:label'] | str = 'id'*, *is_inkscape_label: bool | None = None*) → ShapeList[Wire | Face]

import_svg

Parameters

- **svg_file** (*Union[str, Path, TextIO]*) – svg file
- **flip_y** (*bool, optional*) – flip objects to compensate for svg orientation. Defaults to True.
- **align** (*Align | tuple[Align, Align] | None, optional*) – alignment of the SVG's viewbox, if None, the viewbox's origin will be at *(0,0,0)*. Defaults to Align.MIN.
- **ignore_visibility** (*bool, optional*) – Defaults to False.
- **label_by** (*str, optional*) – XML attribute to use for imported shapes' *label* property. Defaults to "id". Use *inkscape:label* to read labels set from Inkscape's "Layers and Objects" panel.

Raises

ValueError – unexpected shape type

Returns

objects contained in svg

Return type

ShapeList[*Union*[Wire, Face]]

import_svg_as_buildline_code(*file_name: PathLike | str | bytes*, *precision: int = 6*) → tuple[str, str]

translate_to_buildline_code

Translate the contents of the given svg file into executable build123d/BuildLine code.

Parameters

- **file_name** (*PathLike | str | bytes*) – svg file name
- **precision** (*int*) – # digits to round values to. Defaults to # digits in TOLERANCE

Returns

code, builder instance name

Return type

tuple[*str*, *str*]

1.17.5 3D Importers

import_brep(*file_name: PathLike | str | bytes*) → Shape

Import shape from a BREP file

Parameters

file_name (*Union[PathLike, str, bytes]*) – brep file

Raises

ValueError – file not found

Returns

build123d object

Return type

Shape

import_step(*filename: PathLike | str | bytes*) → Compound

Extract shapes from a STEP file and return them as a Compound object.

Parameters
file_name (*Union[PathLike, str, bytes]*) – file path of STEP file to import

Raises
ValueError – can't open file

Returns
contents of STEP file

Return type
Compound

import_stl(*file_name: ~os.PathLike | str | bytes*, *model_unit: ~build123d.build_enums.Unit = <Unit.MM>*) → Face

Extract shape from an STL file and return it as a Face reference object.

Note that importing with this method and creating a reference is very fast while creating an editable model (with Mesher) may take minutes depending on the size of the STL file.

Parameters

- **file_name** (*Union[PathLike, str, bytes]*) – file path of STL file to import
- **model_unit** (*Unit, optional*) – the default unit used when creating the model. For example, Blender defaults to Unit.M. Defaults to Unit.MM.

Raises

- **ValueError** – Could not import file
- **ValueError** – Invalid model_unit

Returns
STL model

Return type
Face

STL Reconstruction

The *detect_primitives()* helper can be used during STL reconstruction to detect analytic planes, cylinders, and spheres in a mesh-like shape and generate algebra-mode code fragments to aid manual redesign.

See *Tutorial: Reconstructing a Design from an STL* for the full workflow and limitations.

detect_primitives(*mesh: Shape*) → tuple[ShapeList[Face], ShapeList[Face], list[str]]

Detect analytic primitives in a mesh and return faces, leftovers, and code.

This is the user-facing entry point for STL-to-BREP reconstruction. The mesh is indexed first so face geometry and adjacency can be reused throughout the pipeline.

Detection proceeds in stages:

1. High-confidence planes are found first from cleaned proxy faces.
2. Spheres are found next from broad radius-signature classification, connected or sewn regions, local sphere fitting, and region growth.
3. Cylinders are detected from area-grouped sewn regions and local cylinder seeds, then grown, refit, and validated.

4. Remaining coplanar connected components are detected as fallback planes.

Each accepted patch is converted into a build123d Face, unmatched mesh faces are returned as leftovers, and the generated code strings are sorted in the same order as the returned primitives.

3D Mesh Import

Both 3MF and STL import (and export) are provided with the *Mesher* class.

For example:

```
importer = Mesher()
cone, cyl = importer.read("example.3mf")
print(
    f"{importer.mesh_count=}, {importer.vertex_counts=}, {importer.triangle_counts=}"
)
print(f"Imported model unit: {importer.model_unit}")
print(f"{cone.label=}")
print(f"{cone.color.to_tuple()=}")
print(f"{cyl.label=}")
print(f"{cyl.color.to_tuple()=}")
```

```
importer.mesh_count=2, importer.vertex_counts=[66, 52], importer.triangle_counts=[128,␣
↪100]
Imported model unit: Unit.MM
cone.label='blue'
cone.color.to_tuple()=(0.0, 0.0, 1.0, 1.0)
cyl.label='red'
cyl.color.to_tuple()=(1.0, 0.0, 0.0, 1.0)
```

1.18 Advanced Topics

1.18.1 Performance considerations in algebra mode

Creating lots of Shapes in a loop means for every step `fuse` and `clean` will be called. In an example like the below, both functions get slower and slower the more objects are already fused. Overall it takes on an M1 Mac 4.76 sec.

```
diam = 80
holes = Sketch()
r = Rectangle(2, 2)
for loc in GridLocations(4, 4, 20, 20):
    if loc.position.X**2 + loc.position.Y**2 < (diam / 2 - 1.8) ** 2:
        holes += loc * r

c = Circle(diam / 2) - holes
```

One way to avoid it is to use lazy evaluation for the algebra operations. Just collect all objects and then call `fuse` (+) once with all objects and `clean` once. Overall it takes 0.19 sec.

```
r = Rectangle(2, 2)
holes = [
    loc * r
    for loc in GridLocations(4, 4, 20, 20).locations
    if loc.position.X**2 + loc.position.Y**2 < (diam / 2 - 1.8) ** 2
```

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```
]

c = Circle(diam / 2) - holes
```

Another way to leverage the vectorized algebra operations is to add a list comprehension of objects to an empty `Part`, `Sketch` or `Curve`:

```
polygons = Sketch() + [
    loc * RegularPolygon(radius=5, side_count=5)
    for loc in GridLocations(40, 30, 2, 2)
]
```

This again ensures one single `fuse` and `clean` call.

1.18.2 Location arithmetic for algebra mode

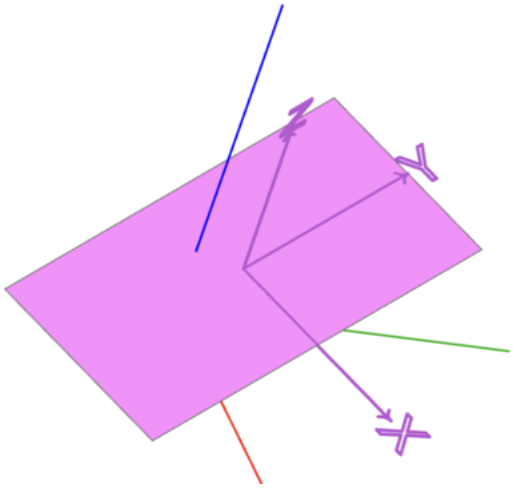
Position a shape relative to the XY plane

For the following use the helper function:

```
def location_symbol(location: Location, scale: float = 1) -> Compound:
    return Compound.make_triad(axes_scale=scale).locate(location)

def plane_symbol(plane: Plane, scale: float = 1) -> Compound:
    triad = Compound.make_triad(axes_scale=scale)
    circle = Circle(scale * .8).edge()
    return (triad + circle).locate(plane.location)
```

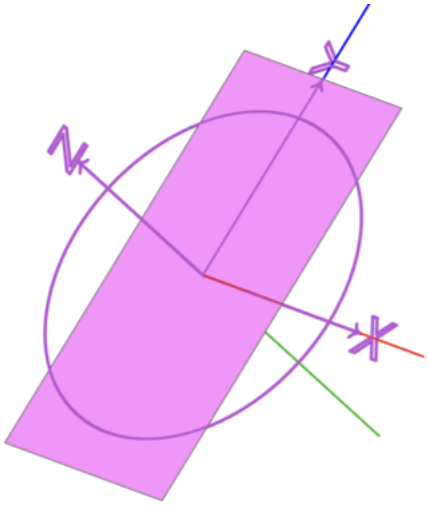
1. **Positioning at a location**

```
loc = Location((0.1, 0.2, 0.3), (10, 20, 30))

face = loc * Rectangle(1, 2)

show_object(face, name="face")
show_object(location_symbol(loc), name="location")
```



2) **Positioning on a plane**

```
plane = Plane.XZ

face = plane * Rectangle(1, 2)

show_object(face, name="face")
show_object(plane_symbol(plane), name="plane")
```

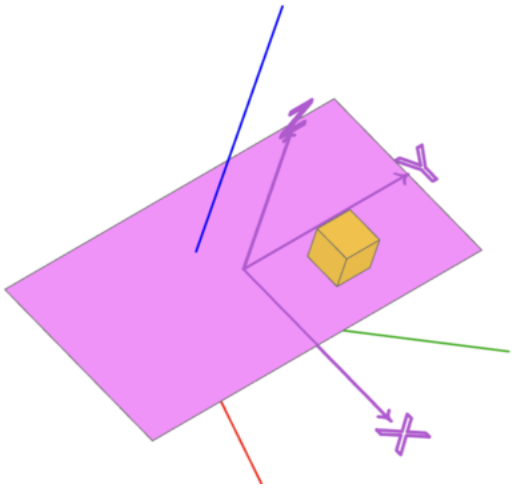


Note: The x-axis and the y-axis of the plane are on the x-axis and the z-axis of the world coordinate system (red and blue axis).

Relative positioning to a plane

1. **Position an object on a plane relative to the plane**

```
loc = Location((0.1, 0.2, 0.3), (10, 20, 30))

face = loc * Rectangle(1,2)

box = Plane(loc) * Pos(0.2, 0.4, 0.1) * Box(0.2, 0.2, 0.2)
# box = Plane(face.location) * Pos(0.2, 0.4, 0.1) * Box(0.2, 0.2, 0.2)
# box = loc * Pos(0.2, 0.4, 0.1) * Box(0.2, 0.2, 0.2)

show_object(face, name="face")
show_object(location_symbol(loc), name="location")
show_object(box, name="box")
```

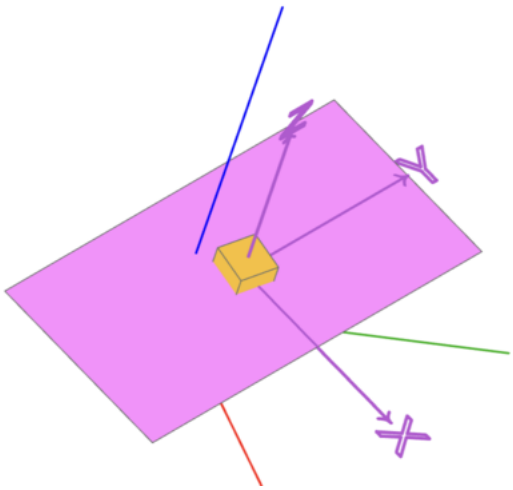


The `X`, `Y`, `Z` components of `Pos(0.2, 0.4, 0.1)` are relative to the `x`-axis, `y`-axis or `z`-axis of the underlying location `loc`.

Note: `Plane(loc) *`, `Plane(face.location) *` and `loc *` are equivalent in this example.

2. **Rotate an object on a plane relative to the plane**

```
loc = Location((0.1, 0.2, 0.3), (10, 20, 30))

face = loc * Rectangle(1,2)

box = Plane(loc) * Rot(Z=80) * Box(0.2, 0.2, 0.2)

show_object(face, name="face")
show_object(location_symbol(loc), name="location")
show_object(box, name="box")
```



The box is rotated via `Rot(Z=80)` around the `z`-axis of the underlying location (and not of the z-axis of the world).

More general:

```
loc = Location((0.1, 0.2, 0.3), (10, 20, 30))
```

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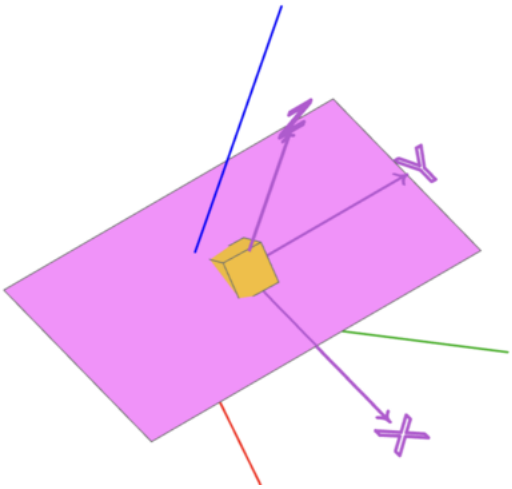
```
face = loc * Rectangle(1,2)

box = loc * Rot(20, 40, 80) * Box(0.2, 0.2, 0.2)

show_object(face, name="face")
show_object(location_symbol(loc), name="location")
show_object(box, name="box")
```



The box is rotated via `Rot(20, 40, 80)` around all three axes relative to the plane.

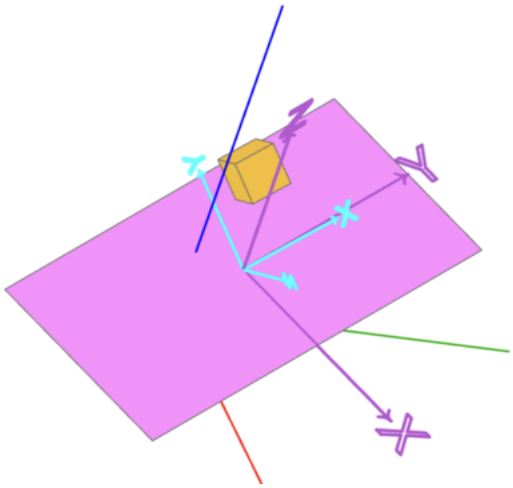
3. **Rotate and position an object relative to a location**

```
loc = Location((0.1, 0.2, 0.3), (10, 20, 30))

face = loc * Rectangle(1,2)

box = loc * Rot(20, 40, 80) * Pos(0.2, 0.4, 0.1) * Box(0.2, 0.2, 0.2)

show_object(face, name="face")
show_object(location_symbol(loc), name="location")
show_object(box, name="box")
show_object(location_symbol(loc * Rot(20, 40, 80), 0.5), options=
↪{"color":(0, 255, 255)}, name="local_location")
```



The box is positioned via `Pos(0.2, 0.4, 0.1)` relative to the location `loc * Rot(20, 40, 80)`

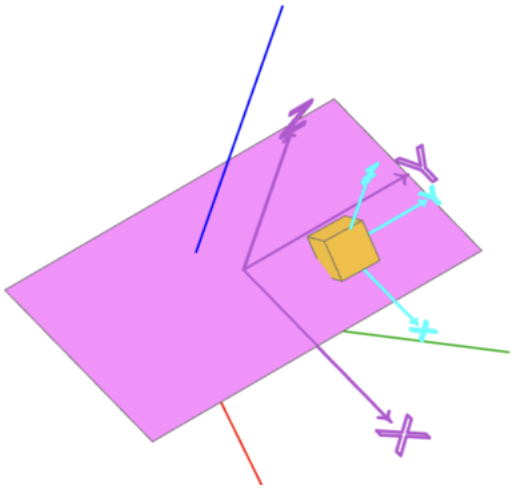
4. **Position and rotate an object relative to a location**

```
loc = Location((0.1, 0.2, 0.3), (10, 20, 30))

face = loc * Rectangle(1,2)

box = loc * Pos(0.2, 0.4, 0.1) * Rot(20, 40, 80) * Box(0.2, 0.2, 0.2)

show_object(face, name="face")
show_object(location_symbol(loc), name="location")
show_object(box, name="box")
show_object(location_symbol(loc * Pos(0.2, 0.4, 0.1), 0.5), options=
↪{"color":(0, 255, 255)}, name="local_location")
```



Note: This is the same as `box = loc * Location((0.2, 0.4, 0.1), (20, 40, 80)) * Box(0.2, 0.2, 0.2)`

1.18.3 Algebraic definition

Objects and arithmetic

Set definitions:

C^3 is the set of all `Part` objects `p` with `p._dim = 3`

C^2 is the set of all `Sketch` objects `s` with `s._dim = 2`

C^1 is the set of all `Curve` objects `c` with `c._dim = 1`

Neutral elements:

c_0^3 is the empty `Part` object `p0 = Part()` with `p0._dim = 3` and `p0.wrapped = None`

c_0^2 is the empty `Sketch` object `s0 = Sketch()` with `s0._dim = 2` and `s0.wrapped = None`

c_0^1 is the empty `Curve` object `c0 = Curve()` with `c0._dim = 1` and `c0.wrapped = None`

Sets of predefined basic shapes:

$B^3 :=$ { `Part`, `Box`, `Cylinder`, `Cone`, `Sphere`, `Torus`, `Wedge`, `Hole`, `CounterBoreHole`, `CounterSinkHole` }

$B^2 :=$ { `Sketch`, `Rectangle`, `Circle`, `Ellipse`, `Rectangle`, `Polygon`, `RegularPolygon`, `Text`, `Trapezoid`, `SlotArc`, `SlotCenterPoint`, `SlotCenterToCenter`, `SlotOverall` }

$B^1 :=$ { `Curve`, `Bezier`, `FilletPolyline`, `PolarLine`, `Polyline`, `Spline`, `Helix`, `CenterArc`, `EllipticalCenterArc`, `ParabolicCenterArc`, `HyperbolicCenterArc`, `RadiusArc`, `SagittaArc`, `TangentArc`, `ThreePointArc`, `JernArc` }

with $B^3 \subset C^3$, $B^2 \subset C^2$ and $B^1 \subset C^1$

Operations:

$+ : C^n \times C^n \to C^n$ with $(a, b) \mapsto a + b$, for $n = 1, 2, 3$

$a + b :=$ `a.fuse(b)` for each operation

$- : C^n \to C^n$ with $a \mapsto -a$, for $n = 1, 2, 3$

$b + (-a) :=$ `b.cut(a)` for each operation (implicit definition)

$\& : C^n \times C^n \to C^n$ with $(a, b) \mapsto a \,\&\, b$, for $n = 2, 3$

$a \,\&\, b :=$ `a.intersect(b)` for each operation

- $\&$ is not defined for $n = 1$ in build123d
- The following relationship holds: $a \,\&\, b = (a + b) + -(a + (-b)) + -(b + (-a))$

Abelian groups

$(C^n,\ c_0^n,\ +,\ -)$ are abelian groups for $n = 1, 2, 3$.

- The implementation `a - b = a.cut(b)` needs to be read as $a + (-b)$ since the group does not have a binary - operation. As such, $a - (b - c) = a + -(b + -c)) \neq a - b + c$
- This definition also includes that neither - nor & are commutative.

Locations, planes and location arithmetic

Set definitions:

$L :=$ { `Location((x, y, z), (a, b, c))` : $x, y, z \in R \wedge a, b, c \in R$}

with a, b, c being angles in degrees.

$P :=$ { `Plane(o, x, z)` : $o, x, zR^3 \wedge \|x\| = \|z\| = 1$}

with `o` being the origin and `x`, `z` the x- and z-direction of the plane.

Neutral element: $l_0 \in L$: `Location()`

Operations:

$* : L \times L \to L$ with $(l_1, l_2) \mapsto l_1 * l_2$

$l_1 * l_2 :=$ `l1 * l2` (multiply two locations)

$* : P \times L \to P$ with $(p, l) \mapsto p * l$

$p * l :=$ `Plane(p.location * l)` (move plane $p \in P$ to location $l \in L$)

Inverse element: $l^{-1} \in L$: `l.inverse()`

Placing objects onto planes

$* : P \times C^n \to C^n$ with $(p, c) \mapsto p * c$, for $n = 1, 2, 3$

Locate an object $c \in C^n$ onto plane $p \in P$, i.e. `c.moved(p.location)`

Placing objects at locations

$* : L \times C^n \to C^n$ with $(l, c) \mapsto l * c$, for $n = 1, 2, 3$

Locate an object $c \in C^n$ at location $l \in L$, i.e. `c.moved(l)`

1.18.4 CAD Object Centers

Finding the center of a CAD object is a surprisingly complex operation. To illustrate let's consider two examples: a simple isosceles triangle and a curved line (their bounding boxes are shown with dashed lines):

One can see that there is are significant differences between the different types of centers. To allow the designer to choose the center that makes the most sense for the given shape there are three possible values for the *CenterOf* Enum:

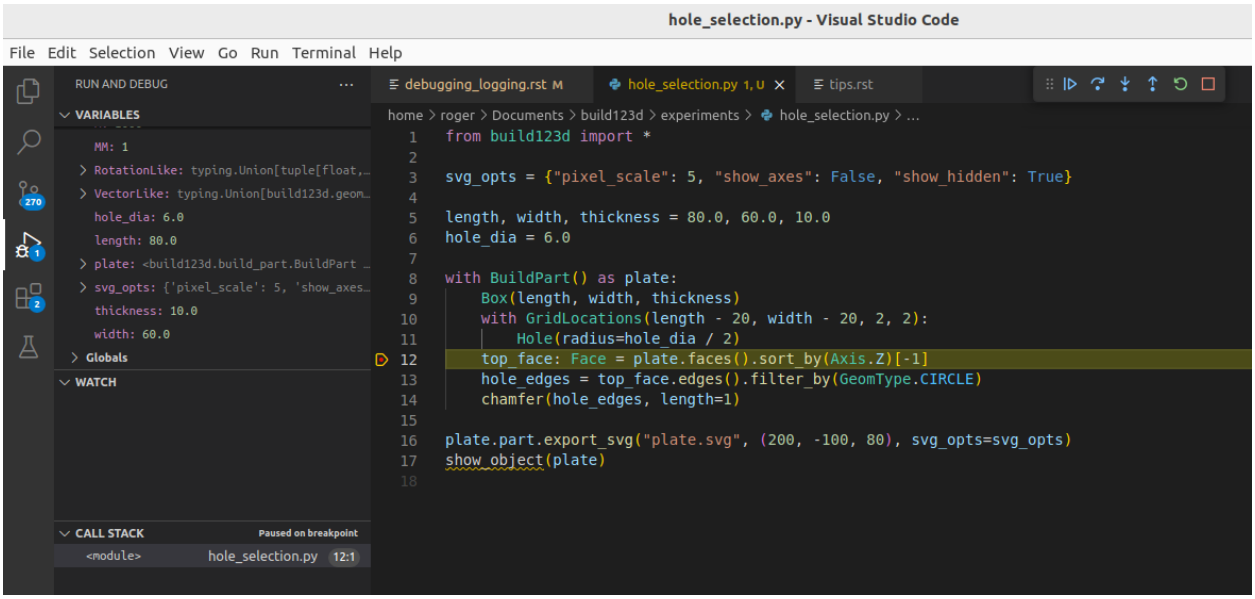
CenterOf	Symbol	1D	2D	3D	Compound
CenterOf.BOUNDING_BOX		✓	✓	✓	✓
CenterOf.GEOMETRY		✓	✓		
CenterOf.MASS		✓	✓	✓	✓

1.18.5 Debugging & Logging

Debugging problems with your build123d design involves the same techniques one would use to debug any Python source code; however, there are some specific techniques that might be of assistance. The following sections describe these techniques.

Python Debugger

Many Python IDEs have step by step debugging systems that can be used to walk through your code monitoring its operation with full visibility of all Python objects. Here is a screenshot of the Visual Studio Code debugger in action:



This shows that a break-point has been encountered and the code operation has been stopped. From here all of the Python variables are visible and the system is waiting on input from the user on how to proceed. One can enter the code that assigns `top_face` by pressing the down arrow button on the top right. Following code execution like this is a very powerful debug technique.

Logging

Build123d support standard python logging and generates its own log stream. If one is using **cq-editor** as a display system there is a built in `Log viewer` tab that shows the current log stream - here is an example of a log stream:

```
[18:43:44.678646] INFO: Entering BuildPart with mode=Mode.ADD which is in different␣
↪scope as parent
[18:43:44.679233] INFO: WorkplaneList is pushing 1 workplanes: [Plane(o=(0.00, 0.00, 0.
↪00), x=(1.00, 0.00, 0.00), z=(0.00, 0.00, 1.00))]
[18:43:44.679888] INFO: LocationList is pushing 1 points: [(p=(0.00, 0.00, 0.00), o=(-0.
↪00, 0.00, -0.00))]
[18:43:44.681751] INFO: BuildPart context requested by Box
[18:43:44.685950] INFO: Completed integrating 1 object(s) into part with Mode=Mode.ADD
[18:43:44.690072] INFO: GridLocations is pushing 4 points: [(p=(-30.00, -20.00, 0.00),␣
↪o=(-0.00, 0.00, -0.00)), (p=(-30.00, 20.00, 0.00), o=(-0.00, 0.00, -0.00)), (p=(30.00,␣
↪-20.00, 0.00), o=(-0.00, 0.00, -0.00)), (p=(30.00, 20.00, 0.00), o=(-0.00, 0.00, -0.
↪00))]
[18:43:44.691604] INFO: BuildPart context requested by Hole
[18:43:44.724628] INFO: Completed integrating 4 object(s) into part with Mode=Mode.
↪SUBTRACT
[18:43:44.728681] INFO: GridLocations is popping 4 points
[18:43:44.747358] INFO: BuildPart context requested by chamfer
[18:43:44.762429] INFO: Completed integrating 1 object(s) into part with Mode=Mode.
↪REPLACE
[18:43:44.765380] INFO: LocationList is popping 1 points
[18:43:44.766106] INFO: WorkplaneList is popping 1 workplanes
[18:43:44.766729] INFO: Exiting BuildPart
```

The build123d logger is defined by:

```
logging.getLogger("build123d").addHandler(logging.NullHandler())
logger = logging.getLogger("build123d")
```

To export logs to a file, the following configuration is recommended:

```
logging.basicConfig(
    filename="myapp.log",
    level=logging.INFO,
    format="%(name)s-%(levelname)s %(asctime)s - [%(filename)s:%(lineno)s - \
    %(funcName)20s() ] - %(message)s",
)
```

Logs can be easily placed in your code - here is an example:

```
logger.info("Exiting %s", type(self).__name__)
```

Printing

Sometimes the best debugging aid is just placing a print statement in your code. Many of the build123d classes are setup to provide useful information beyond their class and location in memory, as follows:

```
plane = Plane.XY.offset(1)
print(f"{plane=}")
```

```
plane=Plane(o=(0.00, 0.00, 1.00), x=(1.00, 0.00, 0.00), z=(0.00, 0.00, 1.00))
```

which shows the origin, x direction, and z direction of the plane.

1.19 Cheat Sheet

Stateful Contexts

BuildLine *BuildPart* *BuildSketch*
GridLocations *HexLocations* *Locations* *PolarLocations*

Objects 1D - BuildLine

Airfoil
ArcArcTangentArc
ArcArcTangentLine
Bezier
BlendCurve
BSpline
CenterArc
ConstrainedArcs
ConstrainedLines
DoubleTangentArc
EllipticalCenterArc
ParabolicCenterArc
HyperbolicCenterArc
FilletPolyline
Helix

IntersectingLine
JernArc
Line
PointArcTangentArc
PointArcTangentLine
PolarLine
Polyline
RadiusArc
SagittaArc
Spline
TangentArc
ThreePointArc

2D - BuildSketch

Arrow
ArrowHead
Circle
DimensionLine
Ellipse
ExtensionLine
Polygon
Rectangle
RectangleRounded
RegularPolygon
SlotArc
SlotCenterPoint
SlotCenterToCenter
SlotOverall
Text
TechnicalDrawing
Trapezoid
Triangle

3D - BuildPart

Box
Cone
ConvexPolyhedron
CounterBoreHole
CounterSinkHole
Cylinder
Hole
Sphere
Torus
Wedge

Operations 1D - BuildLine

add()
bounding_box()

mirror()
offset()
project()
scale()
split()

2D - BuildSketch

add()
chamfer()
fillet()
full_round()
make_face()
make_hull()
mirror()
offset()
project()
scale()
split()
sweep()
trace()

3D - BuildPart

add()
chamfer()
draft()
extrude()
fillet()
loft()
make_brake_formed()
mirror()
offset()
project()
revolve()
scale()
section()
split()
sweep()

Selectors 1D - BuildLine

vertices()
edges()
wires()

2D - BuildSketch

vertices()
edges()
wires()
faces()

3D - BuildPart

vertices()
edges()
wires()
faces()
solids()

Selector Operators

Operator	Operand	Method
>	*Axis*, *Edge*, *Wire*, *SortBy*	*sort_by()*
<	*Axis*, *Edge*, *Wire*, *SortBy*	*sort_by()*
>>	*Axis*, *Edge*, *Wire*, *SortBy*	*group_by()*[-1]
<<	*Axis*, *Edge*, *Wire*, *SortBy*	*group_by()*[0]
|	*Axis*, *Plane*, *GeomType*	*filter_by()*
[]		python indexing / slicing
	Axis	*filter_by_position()*

Edge and Wire Operators

Operator	Operand	Method	Description
@	0.0 <= float <= 1.0	*position_at()*	Position as Vector along object
%	0.0 <= float <= 1.0	*tangent_at()*	Tangent as Vector along object
^	0.0 <= float <= 1.0	*location_at()*	Location along object

Shape Operators

Operator	Operand	Method	Description
==	Any	*is_same()*	Compare CAD objects not including meta data
+	Shape | Iterable[Shape]		Add CAD objects
-	Shape | Iterable[Shape]		Subtract CAD objects
&	Shape | Iterable[Shape]		Intersect CAD objects

Plane Operators

Operator	Operand	Description
==	*Plane*	Check for equality
!=	*Plane*	Check for inequality
-	*Plane*	Reverse direction of normal
*	*Location* | Shape	Relocate

Location Operators

Operator	Operand	Description
==	*Location*	Check for equality
!=	*Location*	Check for inequality
-	*Location*	Reverse direction of normal
&	*Axis* | *Location* | *Plane* | VectorLike | *Shape*	Intersect
*	Shape | *Location* | Iterable[*Location*]	Relocate

Vector Operators

Operator	Operand	Method	Description
+	*Vector*	*add()*	add
-	*Vector*	*sub()*	subtract
*	float	*multiply()*	multiply by scalar
/	float	*multiply()*	divide by scalar

Vertex Operators

Operator	Operand	Method
+	*Vertex*	add()
-	*Vertex*	sub()

Enums

Align	MIN, CENTER, MAX
ApproxOpti	ARC, NONE, SPLINE
AngularDir	CLOCKWISE, COUNTER_CLOCKWISE
CenterOf	GEOMETRY, MASS, BOUNDING_BOX
Extrinsic	XYZ, XZY, YZX, YXZ, ZXY, ZYX, XYX, XZX, YZY, YXY, ZXZ, ZYZ
FontStyle	REGULAR, BOLD, BOLDITALIC, ITALIC
FrameMetho	CORRECTED, FRENET
GeomType	BEZIER, BSPLINE, CIRCLE, CONE, CYLINDER, ELLIPSE, EXTRUSION, HYPERBOLA, LINE, OFFSET, OTHER, PARABOLA, PLANE, REVOLUTION, SPHERE, TORUS
Intrinsic	XYZ, XZY, YZX, YXZ, ZXY, ZYX, XYX, XZX, YZY, YXY, ZXZ, ZYZ
HeadType	CURVED, FILLETED, STRAIGHT
Keep	ALL, TOP, BOTTOM, BOTH, INSIDE, OUTSIDE
Kind	ARC, INTERSECTION, TANGENT
LengthMode	DIAGONAL, HORIZONTAL, VERTICAL
MeshType	OTHER, MODEL, SUPPORT, SOLIDSUPPORT
Mode	ADD, SUBTRACT, INTERSECT, REPLACE, PRIVATE
NumberDisp	DECIMAL, FRACTION
PageSize	A0, A1, A2, A3, A4, A5, A6, A7, A8, A9, A10, LEDGER, LEGAL, LETTER
PositionMo	LENGTH, PARAMETER
PrecisionM	LEAST, AVERAGE, GREATEST, SESSION
Select	ALL, LAST, NEW
Side	BOTH, LEFT, RIGHT
SortBy	LENGTH, RADIUS, AREA, VOLUME, DISTANCE
TextAlign	BOTTOM, CENTER, LEFT, RIGHT, TOP, TOPFIRSTLINE
Transition	RIGHT, ROUND, TRANSFORMED
Unit	MC, MM, CM, M, IN, FT
Until	FIRST, LAST, NEXT, PREVIOUS

1.20 External Tools and Libraries

The following sections describe tools and libraries external to build123d that extend its functionality.

1.20.1 Editors & Viewers

ocp-vscode

A viewer for OCP based Code-CAD (CadQuery, build123d) integrated into VS Code.

See: ocp-vscode (formerly known as cq_vscode)

Watch Jern create three build123d designs in realtime with Visual Studio Code and the ocp-vscode viewer extension in a timed event from the TooTallToby 2024 Spring Open Tournament: build123d entry video

cq-editor fork

GUI editor based on PyQT. This fork has changes from jdegenstein to allow easier use with build123d.

See: jdegenstein's fork of cq-editor

Yet Another CAD Viewer

A web-based CAD viewer for OCP models (CadQuery/build123d) that runs in any modern browser and supports static site deployment. Features include interactive inspection of faces, edges, and vertices, measurement tools, per-model clipping planes, transparency control, and hot reloading via `yacv-server`. It also has a build123d playground for editing and sharing models directly in the browser (demo).

See: Yet Another CAD Viewer

PartCAD VS Code extension

A wrapper around `ocp-vscode` (see above) which requires build123d scripts to be packaged using `PartCAD` (see below). While it requires the overhead of maintaining the package, it provides some convenience features (such as UI controls to export models) as well as functional features (such as UI controls to pass parameters into build123d scripts and AI-based generative design tools).

It's also the most convenient tool to create new packages and parts. More PDM and PLM features are expected to arrive soon.

1.20.2 Part Libraries

bd_warehouse

On-demand generation of parametric parts that seamlessly integrate into build123d projects.

Parts available include:

- fastener - Nuts, Screws, Washers and custom holes
- flange - Standardized parametric flanges
- pipe - Standardized parametric pipes
- thread - Parametric helical threads (Iso, Acme, Plastic, etc.)

See: bd_warehouse

bd_beams_and_bars

2D sections and 3D beams generation (UPN, IPN, UPE, flat bars, ...)

See: bd_beams_and_bars

Superellipses & Superellipsoids

Superellipses are a more sophisticated alternative to rounded rectangles, with smoothly changing curvature. They are flexible shapes that can be adjusted by changing the "exponent" to get a result that varies between rectangular and elliptical, or from square, through squircle, to circle, and beyond...

Superellipses can be found:

- in typefaces such as Melior, Eurostyle, and Computer Modern
- as the shape of airliner windows, tables, plates
- clipping the outline of iOS app icons

They were named and popularized in the 1950s-1960s by the Danish mathematician and poet Piet Hein, who used them in the winning design for the Sergels Torg roundabout in Stockholm.

See: Superellipses & Superellipsoids

Public PartCAD repository

See partcad.org for all the models packaged and published using `PartCAD` (see below). This repository contains individual parts, as well as large assemblies created using those parts. See the OpenVMP robot as an example of an assembly

py_gearworks generator

A gear generation framework that allows easy creation of a wide range of gears and drives.

See py_gearworks

bd_vslot

A library of V-Slot linear rail components, including V-Slot rails.

See: bd_vslot

1.20.3 Tools

blendquery

CadQuery and build123d integration for Blender.

See: blendquery

nething

3D generative AI for CAD modeling. Now everyone is an engineer. Make your ideas real.

See: nething

Listen to the following podcast which discusses **nething** in detail: The Next Byte Podcast

ocp-freecad-cam

CAM for CadQuery and Build123d by leveraging FreeCAD library. Visualizes in CQ-Editor and ocp-cad-viewer. Spiritual successor of cq-cam

See: ocp-freecad-cam

PartCAD

A package manager for CAD models. Build123d is the most supported Code-CAD framework, but CadQuery and OpenSCAD are also supported. It can be used by build123d designs to import parts from PartCAD repositories, and to publish build123d designs to be consumed by others.

MakerRepo library (mr)

The `makerrepo` Python package (imported as `mr`) is a lightweight library that provides decorators such as `@artifact`, `@customizable`, and `@cached` to annotate functions that build your models. The decorators have no effect on your existing build123d code until it is discovered and run by tools such as the makerrepo-cli or MakerRepo.com CI. The goal is to enable a code-driven workflow locally (e.g. command-line tools) or in CI. The library does not assume how it will be consumed, so annotated functions can be used with other tools and frameworks as well.

See MakerRepo Library Docs for more information and LaunchPlatform/makerrepo for source code.

makerrepo-cli

Command-line tool (available as `makerrepo-cli` or `mr`) to build artifacts, run generators, snapshot artifacts, and manage cache locally. It scans the current directory for Python packages and modules that use the MakerRepo library decorators.

See MakerRepo CLI for documentation and LaunchPlatform/makerrepo-cli for source code.

dl4to4ocp

Library that helps perform topology optimization on your OCP-based CAD models (CadQuery/Build123d/...) using the dl4to library.

See: dl4to4ocp

OCP.wasm

This project ports the low-level dependencies required for build123d to run in a browser. For a fully featured frontend, check out `Yet Another CAD Viewer` (see above).

See: OCP.wasm

partomatic

Partomatic provides a standardized system for building parametric models in build123d. The open nature of build123d is its strength, but it makes it difficult to build standardized tooling to interface with your projects. It makes it easy to:

- import and export configuration files
- easily export models for projects that provide large numbers of intersectional options
- share the built-in web interface allowing end-users to change properties and see the results quickly
- generate logs for compilation and web interface events that can be consumed by an OpenTelemetry platform

See: Partomatic

1.21 Builder Common API Reference

The following are common to all the builders.

1.21.1 Selector Methods

`Builder.vertices`(*select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Vertex]

Return Vertices

Return either all or the vertices created during the last operation.

Parameters
`select` (`Select, optional`) – Vertex selector. Defaults to Select.ALL.

Returns
Vertices extracted

Return type
ShapeList[Vertex]

`Builder.faces`(*select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Face]

Return Faces

Return either all or the faces created during the last operation.

Parameters
select (Select, *optional*) – Face selector. Defaults to Select.ALL.

Returns
Faces extracted

Return type
ShapeList[Face]

Builder.**edges**(*select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Edge]

Return Edges

Return either all or the edges created during the last operation.

Parameters
select (Select, *optional*) – Edge selector. Defaults to Select.ALL.

Returns
Edges extracted

Return type
ShapeList[Edge]

Builder.**wires**(*select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Wire]

Return Wires

Return either all or the wires created during the last operation.

Parameters
select (Select, *optional*) – Wire selector. Defaults to Select.ALL.

Returns
Wires extracted

Return type
ShapeList[Wire]

Builder.**solids**(*select: ~build123d.build_enums.Select = <Select.ALL>*) → ShapeList[Solid]

Return Solids

Return either all or the solids created during the last operation.

Parameters
select (Select, *optional*) – Solid selector. Defaults to Select.ALL.

Returns
Solids extracted

Return type
ShapeList[Solid]

1.21.2 Enums

class Align(*value*)

Align object about Axis

CENTER = 2

MAX = 3

MIN = 1

NONE = None

class CenterOf(*value*)

Center Options

BOUNDING_BOX = 3

GEOMETRY = 1

MASS = 2

class FontStyle(*value*)

Text Font Styles

BOLD = 2

BOLDITALIC = 4

ITALIC = 3

REGULAR = 1

class GeomType(*value*)

CAD geometry object type

BEZIER = 6

BSPLINE = 7

CIRCLE = 12

CONE = 3

CYLINDER = 2

ELLIPSE = 13

EXTRUSION = 9

HYPERBOLA = 14

LINE = 11

OFFSET = 10

OTHER = 16

PARABOLA = 15

PLANE = 1

REVOLUTION = 8

SPHERE = 4

TORUS = 5

class Keep(*value*)

Split options

ALL = 1

BOTH = 3

BOTTOM = 2

INSIDE = 4

OUTSIDE = 5

TOP = 6

class Kind(*value*)

Offset corner transition

ARC = 1

INTERSECTION = 2

TANGENT = 3

class Mode(*value*)

Combination Mode

ADD = 1

INTERSECT = 3

PRIVATE = 5

REPLACE = 4

SUBTRACT = 2

class Select(*value*)

Selector scope - all, last operation or new objects

ALL = 1

LAST = 2

NEW = 3

class SortBy(*value*)

Sorting criteria

AREA = 3

DISTANCE = 5

LENGTH = 1

RADIUS = 2

VOLUME = 4

class Transition(*value*)

Sweep discontinuity handling option

RIGHT = 1

ROUND = 2

TRANSFORMED = 3

class Until(*value*)

Extrude limit

FIRST = 4

LAST = 2

NEXT = 1

PREVIOUS = 3

1.21.3 Locations

class Locations(**pts: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | Vertex | Location | Face | Plane | Axis | Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | Vertex | Location | Face | Plane | Axis]*)

Location Context: Push Points

Creates a context of locations for Part or Sketch

Parameters

pts (*Union[VectorLike, Vertex, Location, Face, Plane, Axis] or iterable of same*) – sequence of points to push

Variables

local_locations (*list{Location}*) – locations relative to workplane

local_locations

values independent of workplanes

class GridLocations(*x_spacing: float*, *y_spacing: float*, *x_count: int*, *y_count: int*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>)*)

Location Context: Rectangular Array

Creates a context of rectangular array of locations for Part or Sketch

Parameters

- **x_spacing** (*float*) – horizontal spacing
- **y_spacing** (*float*) – vertical spacing
- **x_count** (*int*) – number of horizontal points
- **y_count** (*int*) – number of vertical points
- **align** (*Union[Align, tuple[Align, Align]], optional*) – align min, center, or max of object. Defaults to (Align.CENTER, Align.CENTER).

Variables

- **x_spacing** (*float*) – horizontal spacing
- **y_spacing** (*float*) – vertical spacing
- **x_count** (*int*) – number of horizontal points
- **y_count** (*int*) – number of vertical points

- **align** (*Union[Align, tuple[Align, Align]]*) – align min, center, or max of object.
- **local_locations** (*list{Location}*) – locations relative to workplane

Raises

ValueError – Either x or y count must be greater than or equal to one.

local_locations

values independent of workplanes

max

top right corner

min

bottom left corner

size

size of the grid

class HexLocations(*radius: float*, *x_count: int*, *y_count: int*, *major_radius: bool = False*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] = (<Align.CENTER>, <Align.CENTER>)*)

Location Context: Hex Array

Creates a context of hexagon array of locations for Part or Sketch. When creating hex locations for an array of circles, set *radius* to the radius of the circle plus one half the spacing between the circles.

Parameters

- **radius** (*float*) – distance from origin to vertices (major), or optionally from the origin to side (minor or apothem) with major_radius = False
- **x_count** (*int*) – number of points (> 0)
- **y_count** (*int*) – number of points (> 0)
- **major_radius** (*bool*) – If True the radius is the major radius, else the radius is the minor radius (also known as inscribed radius). Defaults to False.
- **align** (*Union[Align, tuple[Align, Align]], optional*) – align min, center, or max of object. Defaults to (Align.CENTER, Align.CENTER).

Variables

- **radius** (*float*) – distance from origin to vertices (major), or optionally from the origin to side (minor or apothem) with major_radius = False
- **apothem** (*float*) – radius of the inscribed circle, also known as minor radius
- **x_count** (*int*) – number of points (> 0)
- **y_count** (*int*) – number of points (> 0)
- **major_radius** (*bool*) – If True the radius is the major radius, else the radius is the minor radius (also known as inscribed radius).
- **align** (*Union[Align, tuple[Align, Align]]*) – align min, center, or max of object.
- **diagonal** (*float*) – major radius
- **local_locations** (*list{Location}*) – locations relative to workplane

Raises

ValueError – Spacing and count must be > 0

local_locations

values independent of workplanes

class PolarLocations(*radius: float*, *count: int*, *start_angle: float = 0.0*, *angular_range: float = 360.0*, *rotate: bool = True*, *endpoint: bool = False*)

Location Context: Polar Array

Creates a context of polar array of locations for Part or Sketch

Parameters

- **radius** (*float*) – array radius
- **count** (*int*) – Number of points to push
- **start_angle** (*float, optional*) – angle to first point from +ve X axis. Defaults to 0.0.
- **angular_range** (*float, optional*) – magnitude of array from start angle. Defaults to 360.0.
- **rotate** (*bool, optional*) – Align locations with arc tangents. Defaults to True.
- **endpoint** (*bool, optional*) – If True, *start_angle* + *angular_range* is the last sample. Otherwise, it is not included. Defaults to False.

Variables

local_locations (*list{Location}*) – locations relative to workplane

Raises

ValueError – Count must be greater than or equal to 1

local_locations

values independent of workplanes

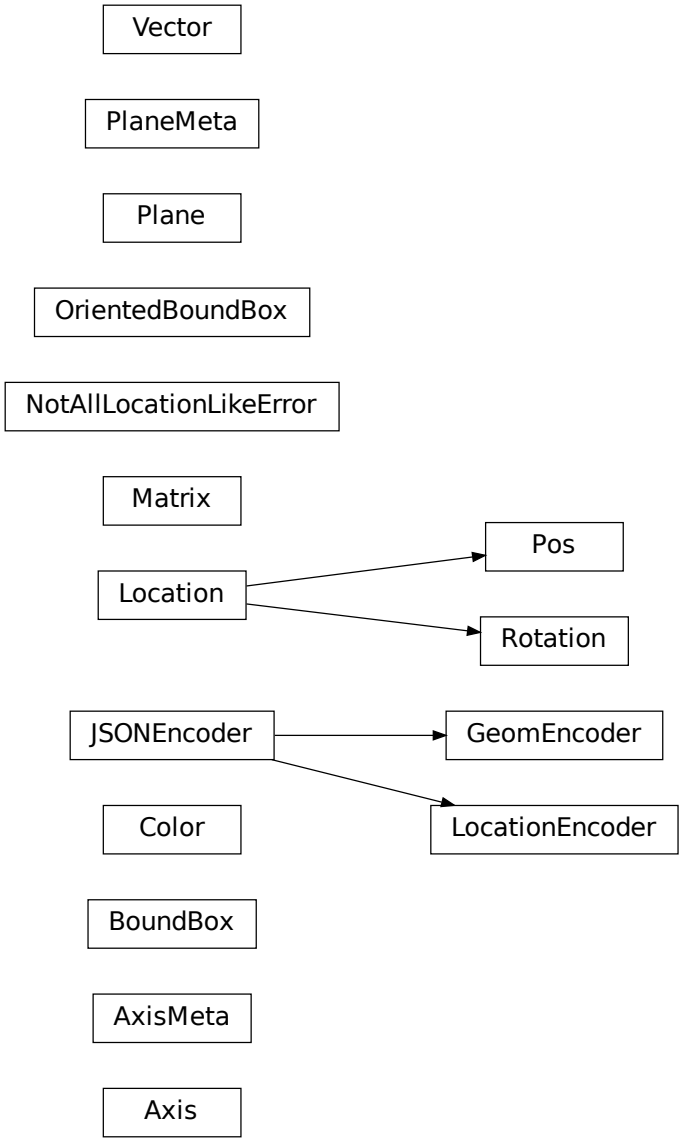
1.22 Direct API Reference

The Direct API is an interface layer between the primary user interface API (the Builders) and the OpenCascade (OCCT) API. This API is based on the CadQuery Direct API (thank you to all of the CadQuery contributors that made this possible) with the following major changes:

- PEP8 compliance
- New Axis class
- New ShapeList class enabling sorting and filtering of shape objects
- Literal strings replaced with Enums

1.22.1 Geometric Objects

The geometric classes defined by build123d are defined below. This parameters to the CAD objects described in the following section are frequently of these types.



class Axis(**args: Any*, ***kwargs: Any*)

Axis defined by point and direction or by two points

Parameters

- **origin** (*VectorLike*) – start point
- **direction** (*VectorLike*) – direction
- **end_point** (*VectorLike*) – point used with origin to define direction
- **edge** (Edge) – origin & direction defined by start of edge
- **location** (Location) – location to convert to axis

Variables

- **position** (Vector) – the global position of the axis origin
- **direction** (Vector) – the normalized direction vector
- **wrapped** (*gp_Ax1*) – the OCP axis object

__copy__() → *Axis*

Return copy of self

__deepcopy__(*_memo*) → *Axis*

Return deepcopy of self

__neg__() → *Axis*

Flip direction operator -

angle_between(*other:* Axis) → float

calculate angle between axes

Computes the angular value, in degrees, between the direction of self and other between 0° and 360°.

Parameters
other (Axis) – axis to compare to

Returns
angle between axes

Return type
float

property direction: *Vector*

The normalized direction of the Axis

intersect(**args*, ***kwargs*)

Find intersection of axis and geometric object or shape

is_coaxial(*other:* Axis, *angular_tolerance: float = 1e-05*, *linear_tolerance: float = 1e-05*) → bool

are axes coaxial

True if the angle between self and other is lower or equal to angular_tolerance and the distance between self and other is lower or equal to linear_tolerance.

Parameters

- **other** (Axis) – axis to compare to
- **angular_tolerance** (*float, optional*) – max angular deviation. Defaults to 1e-5.
- **linear_tolerance** (*float, optional*) – max linear deviation. Defaults to 1e-5.

Returns
axes are coaxial

Return type
bool

is_normal(*other:* Axis, *angular_tolerance: float = 1e-05*) → bool

are axes normal

Returns True if the direction of this and another axis are normal to each other. That is, if the angle between the two axes is equal to 90° within the angular_tolerance.

Parameters

- **other** (Axis) – axis to compare to
- **angular_tolerance** (*float, optional*) – max angular deviation. Defaults to 1e-5.

Returns
axes are normal

Return type
bool

is_opposite(*other:* Axis, *angular_tolerance: float = 1e-05*) → bool

are axes opposite

Returns True if the direction of this and another axis are parallel with opposite orientation. That is, if the angle between the two axes is equal to 180° within the angular_tolerance.

Parameters

- **other** (Axis) – axis to compare to
- **angular_tolerance** (*float, optional*) – max angular deviation. Defaults to 1e-5.

Returns
axes are opposite

Return type
bool

is_parallel(*other:* Axis, *angular_tolerance: float = 1e-05*) → bool

are axes parallel

Returns True if the direction of this and another axis are parallel with same orientation or opposite orientation. That is, if the angle between the two axes is equal to 0° or 180° within the angular_tolerance.

Parameters

- **other** (Axis) – axis to compare to
- **angular_tolerance** (*float, optional*) – max angular deviation. Defaults to 1e-5.

Returns
axes are parallel

Return type
bool

is_skew(*other:* Axis, *tolerance: float = 1e-05*) → bool

are axes skew

Returns True if this axis and another axis are skew, meaning they are neither parallel nor coplanar. Two axes are skew if they do not lie in the same plane and never intersect.

Mathematically, this means:

- The axes are **not parallel** (the cross product of their direction vectors is nonzero).
- The axes are **not coplanar** (the vector between their positions is not aligned with the plane spanned by their directions).

If either condition is false (i.e., the axes are parallel or coplanar), they are not skew.

Parameters

- **other** (Axis) – axis to compare to
- **tolerance** (*float, optional*) – max deviation. Defaults to 1e-5.

Returns
axes are skew

Return type
bool

`located`(*new_location:* Location)

relocates self to a new location possibly changing position and direction

`property location:` *Location*

Return self as Location

`property position:` *Vector*

The position or origin of the Axis

`reverse`() → *Axis*

Return a copy of self with the direction reversed

`to_plane`() → *Plane*

Return self as Plane

`property wrapped`

OCP object

`class BoundBox`(**args*, ***kwargs*)

A BoundingBox for a Shape

`add`(*obj: tuple[float, float, float] | Vector | BoundBox*, *tol: float | None = None*) → *BoundBox*

Returns a modified (expanded) bounding box

obj can be one of several things:

1. a 3-tuple corresponding to x,y, and z amounts to add
2. a vector, containing the x,y,z values to add
3. another bounding box, where a new box will be created that encloses both.

This bounding box is not changed.

Parameters

- **obj** – tuple[float, float, float] | Vector | BoundBox]:
- **tol** – float: (Default value = None)

Returns:

`center`() → *Vector*

Return center of the bounding box

`property diagonal: float`

body diagonal length (i.e. object maximum size)

`static find_outside_box_2d`(*bb1:* BoundBox, *bb2:* BoundBox) → *BoundBox* | None

Compares bounding boxes

Compares bounding boxes. Returns none if neither is inside the other. Returns the outer one if either is outside the other.

BoundBox.is_inside works in 3d, but this is a 2d bounding box, so it doesn't work correctly plus, there was all kinds of rounding error in the built-in implementation i do not understand.

Parameters

- **bb1** – BoundBox:
- **bb2** – BoundBox:

Returns:

classmethod from_topo_ds(*shape: TopoDS_Shape*, *tolerance: float | None = None*, *optimal: bool = True*) → *BoundBox*

Constructs a bounding box from a TopoDS_Shape

Parameters

- **shape** – TopoDS_Shape:
- **tolerance** – float: (Default value = None)
- **optimal** – bool: This algorithm builds precise bounding box (Default value = True)

Returns:

is_inside(*second_box:* BoundBox) → bool

Is the provided bounding box inside this one?

Parameters
b2 – BoundBox:

Returns:

property measure: float

Return the overall Lebesgue measure of the bounding box.

- For 1D objects: length
- For 2D objects: area
- For 3D objects: volume

overlaps(*other:* BoundBox, *tolerance: float = 1e-06*) → bool

Check if this bounding box overlaps with another.

Parameters

- **other** – BoundBox to check overlap with
- **tolerance** – Distance tolerance for overlap detection

Returns
True if bounding boxes overlap (share any volume), False otherwise

to_align_offset(*align: Align | None | tuple[Align | None, Align | None] | tuple[Align | None, Align | None, Align | None]*) → *Vector*

Amount to move object to achieve the desired alignment

class Color(**args*, ***kwargs*)

Color object based on OCCT Quantity_ColorRGBA.

Variables
wrapped (*Quantity_ColorRGBA*) – the OCP color object

__copy__() → *Color*

Return copy of self

__deepcopy__(*_memo*) → *Color*

Return deepcopy of self

classmethod categorical_set(*color_count: int*, *starting_hue: str | tuple[str, float | int] | tuple[float | int, float | int, float | int] | tuple[float | int, float | int, float | int, float | int] | int | tuple[int, int] | Color | Quantity_ColorRGBA | float = 0.0*, *alpha: float | Iterable[float] = 1.0*) → list[*Color*]

Generate a palette of evenly spaced colors.

Creates a list of visually distinct colors suitable for representing discrete categories (such as different parts, assemblies, or data series). Colors are evenly spaced around the hue circle and share consistent lightness and saturation levels, resulting in balanced perceptual contrast across all hues.

Produces palettes similar in appearance to the **Tableau 10** and **D3 Category10** color sets—both widely recognized standards in data visualization for their clarity and accessibility. These values have been empirically chosen to maintain consistent perceived brightness across hues while avoiding overly vivid or dark colors.

Parameters

- **color_count** (*int*) – Number of colors to generate.
- **starting_hue** (*ColorLike | float*) – Either a Color-like object or a hue value in the range [0.0, 1.0] that defines the starting color.
- **alpha** (*float | Iterable[float]*) – Alpha value(s) for the colors. Can be a single float or an iterable of length *color_count*.

Returns

List of generated colors.

Return type

list[Color]

Raises

ValueError – If starting_hue is out of range or alpha length mismatch.

class Location(**args: Any*, ***kwargs: Any*)

Location in 3D space. Depending on usage can be absolute or relative.

This class wraps the TopLoc_Location class from OCCT. It can be used to move Shape objects in both relative and absolute manner. It is the preferred type to locate objects in build123d.

Variables

wrapped (*TopLoc_Location*) – the OCP location object

__copy__() → *Location*

Lib/copy.py shallow copy

__deepcopy__(*_memo*) → *Location*

Lib/copy.py deep copy

__eq__(*other: object*) → bool

Compare Locations

__mul__(*other:* Location | *Iterable[*Location*]*) → *Location* | list[*Location*]

Combine locations

__neg__() → *Location*

Flip the orientation without changing the position operator -

__pow__(*exponent: int*) → *Location*

center() → *Vector*

Return center of the location - useful for sorting

intersect(**args*, ***kwargs*)

Find intersection of location and geometric object or shape

inverse() → *Location*

Inverted location

mirror(*mirror_plane:* Plane) → *Location*

Return a new Location mirrored across the given plane.

This method reflects both the position and orientation of the current Location across the specified mirror_plane using affine vector mathematics.

Due to the mathematical properties of reflection:

- The true mirror of a right-handed coordinate system is a *left-handed* one.

However, *build123d* requires all coordinate systems to be right-handed. Therefore, this implementation: - Reflects the X and Z directions across the mirror plane - Recomputes the Y direction as: $Y = X \times Z$

This ensures the resulting Location maintains a valid right-handed frame, while remaining as close as possible to the geometric mirror.

Parameters
mirror_plane (Plane) – The plane to mirror across.

Returns
A new mirrored Location that preserves right-handedness.

Return type
Location

property orientation: *Vector*

Extract orientation/rotation component of self

Returns
orientation part of Location

Return type
Vector

property position: *Vector*

Extract Position component of self

Returns
Position part of Location

Return type
Vector

to_axis() → *Axis*

Convert the location into an Axis

to_tuple() → tuple[tuple[float, float, float], tuple[float, float, float]]

Convert the location to a translation, rotation tuple.

property wrapped: **TopLoc_Location**

OCP object

property x_axis: *Axis*

Default X axis when used as a plane

property y_axis: *Axis*

Default Y axis when used as a plane

property z_axis: *Axis*

Default Z axis when used as a plane

class LocationEncoder(*, *skipkeys=False*, *ensure_ascii=True*, *check_circular=True*, *allow_nan=True*, *sort_keys=False*, *indent=None*, *separators=None*, *default=None*)

Custom JSON Encoder for Location values

Example:

```
data_dict = {
    "part1": {
        "joint_one": Location((1, 2, 3), (4, 5, 6)),
        "joint_two": Location((7, 8, 9), (10, 11, 12)),
    },
    "part2": {
        "joint_one": Location((13, 14, 15), (16, 17, 18)),
        "joint_two": Location((19, 20, 21), (22, 23, 24)),
    },
}
json_object = json.dumps(data_dict, indent=4, cls=LocationEncoder)
with open("sample.json", "w") as outfile:
    outfile.write(json_object)
with open("sample.json", "r") as infile:
    copy_data_dict = json.load(infile, object_hook=LocationEncoder.location_hook)
```

default(*o:* Location) → dict

Return a serializable object

static location_hook(*obj*) → dict

Convert Locations loaded from json to Location objects

Example

read_json = json.load(infile, object_hook=LocationEncoder.location_hook)

class Pos(**args*, ***kwargs*)

A position only sub-class of Location

Rot

alias of *Rotation*

class Matrix(**args*, ***kwargs*)

A 3d , 4x4 transformation matrix.

Used to move geometry in space.

The provided “matrix” parameter may be None, a gp_GTrsf, or a nested list of values.

If given a nested list, it is expected to be of the form:

[[m11, m12, m13, m14],
[m21, m22, m23, m24], [m31, m32, m33, m34]]

A fourth row may be given, but it is expected to be: [0.0, 0.0, 0.0, 1.0] since this is a transform matrix.

Variables
wrapped (*gp_GTrsf*) – the OCP transformation function

__copy__() → *Matrix*

Return copy of self

__deepcopy__(*_memo*) → *Matrix*

Return deepcopy of self

inverse() → *Matrix*

Invert Matrix

multiply(*other*)

Matrix multiplication

rotate(*axis:* Axis, *angle: float*)

General rotate about axis by angle in degrees

transposed_list() → Sequence[float]

Needed by the cqparts gltf exporter

class Plane(**args: Any*, ***kwargs: Any*)

A plane is positioned in space with a coordinate system such that the plane is defined by the origin, x_dir (X direction), y_dir (Y direction), and z_dir (Z direction) of this coordinate system, which is the "local coordinate system" of the plane. The z_dir is a vector normal to the plane. The coordinate system is right-handed.

A plane allows the use of local 2D coordinates, which are later converted to global, 3d coordinates when the operations are complete.

Planes can be created from faces as workplanes for feature creation on objects.

Name	x_dir	y_dir	z_dir
XY	+x	+y	+z
YZ	+y	+z	+x
ZX	+z	+x	+y
XZ	+x	+z	-y
YX	+y	+x	-z
ZY	+z	+y	-x
front	+x	+z	-y
back	-x	+z	+y
left	-y	+z	-x
right	+y	+z	+x
top	+x	+y	+z
bottom	+x	-y	-z
isometric	+x+y	-x+y+z	+x+y-z

Parameters

- **gp_pln** (*gp_Pln*) – an OCCT plane object
- **origin** (*tuple[float, float, float] | Vector*) – the origin in global coordinates
- **x_dir** (*tuple[float, float, float] | Vector | None*) – an optional vector representing the X Direction. Defaults to None.

- **y_dir** (*tuple[float, float, float] | Vector | None*) – optional Y direction. Mutually exclusive with z_dir. Requires x_dir.
- **z_dir** (*tuple[float, float, float] | Vector | None*) – the normal direction for the plane. Defaults to (0, 0, 1).

Variables

- **origin** (Vector) – global position of local (0,0,0) point
- **x_dir** (Vector) – x direction
- **y_dir** (Vector) – y direction
- **z_dir** (Vector) – z direction
- **forward_transform** (Matrix) – forward location transformation matrix
- **reverse_transform** (Matrix) – reverse location transformation matrix
- **wrapped** (*gp_Pln*) – the OCP plane object

Raises

- **ValueError** – z_dir must be non null
- **ValueError** – y_dir must be non null
- **ValueError** – x_dir must be non null
- **ValueError** – the specified x_dir is not orthogonal to the provided normal
- **ValueError** – x_dir and y_dir must not be parallel
- **ValueError** – the specified x_dir is not orthogonal to the provided normal

Returns
A plane

Return type
Plane

__copy__() → *Plane*

Return copy of self

__deepcopy__(*_memo*) → *Plane*

Return deepcopy of self

__eq__(*other: object*)

Are planes equal operator ==

__mul__(*other:* Location | Plane | *Iterable[*Location | Plane*]*) → *Location* | list[*Location*]

__neg__() → *Plane*

Reverse z direction of plane operator -

contains(*obj:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]* | Axis, *tolerance: float* = *1e-06*) → bool

Is this point or Axis fully contained in this plane?

Parameters

- **obj** (*VectorLike | Axis*) – point or Axis to evaluate
- **tolerance** (*float, optional*) – comparison tolerance. Defaults to TOLERANCE.

Returns
self contains point or Axis

Return type
bool

property forward_transform

forward location transformation matrix

from_local_coords(*obj: tuple* | Vector | *Any* | BoundBox)

Reposition the object relative from this plane

Parameters

- **obj** – VectorLike | Shape | BoundBox an object to reposition. Note that
- **classes.** (*type Any refers to all topological*)

Returns
an object of the same type, but repositioned to world coordinates

static get_topods_face_normal(*face: TopoDS_Face*) → *Vector*

Find the normal at the center of a TopoDS_Face

intersect(**args*, ***kwargs*)

Find intersection of plane and geometric object or shape

property location: *Location*

Return Location representing the origin and z direction

location_between(*other:* Plane) → *Location*

Return a location representing the translation from self to other

move(*loc:* Location | Plane) → *Plane*

Change the position & orientation of self by applying a relative location

Parameters
loc (Location | Plane) – relative change

Returns
relocated self

Return type
Plane

moved(*loc:* Location | Plane) → *Plane*

Change the position & orientation of a copy of self by applying a relative location

Parameters
loc (Location | Plane) – relative change

Returns
relocated plane

Return type
Plane

offset(*amount: float*) → *Plane*

Move the Plane by amount in the direction of z_dir

property origin: *Vector*

global position of local (0,0,0) point

reverse() → *Plane*

Reverse z direction of plane

property reverse_transform

reverse location transformation matrix

rotated(*rotation:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float] = (0, 0, 0)*, *ordering:* *Extrinsic* | *Intrinsic* | *None = None*) → *Plane*

Returns a copy of this plane, rotated about the specified axes

The origin of the workplane is unaffected by the rotation.

Rotations are done in order x, y, z. If you need a different order, specify ordering. e.g. Intrinsic.ZYX changes rotation to (z angle, y angle, x angle) and rotates in that order.

Parameters

- **rotation** (*VectorLike, optional*) – (x angle, y angle, z angle). Defaults to (0, 0, 0)
- **ordering** (*Intrinsic | Extrinsic, optional*) – order of rotations in Intrinsic or Extrinsic rotation mode. Defaults to Intrinsic.XYZ

Returns
a copy of this plane rotated as requested.

Return type
Plane

shift_origin(*locator:* Axis | *VectorLike* | Vertex) → *Plane*

shift plane origin

Creates a new plane with the origin moved within the plane to the point of intersection of the axis or at the given Vertex. The plane's x_dir and z_dir are unchanged.

Parameters
locator (*Axis | VectorLike | Vertex*) – Either Axis that intersects the new plane origin or Vertex within Plane.

Raises

- **ValueError** – Vertex isn't within plane
- **ValueError** – Point isn't within plane
- **ValueError** – Axis doesn't intersect plane

Returns
plane with new origin

Return type
Plane

to_gp_ax2() → gp_Ax2

Return gp_Ax2 version of the plane

to_gp_ax3() → gp_Ax3

Return gp_Ax3 version of the plane

to_local_coords(*obj:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]* | *Any* | BoundBox)

Reposition the object relative to this plane

Parameters

- **obj** – VectorLike | Shape | BoundBox an object to reposition. Note that
- **classes.** (*type Any refers to all topological*)

Returns
an object of the same type, but repositioned to local coordinates

property wrapped: gp_Pln

The OCP object

property x_dir: *Vector*

Local X direction of the plane.

property y_dir: *Vector*

Local Y direction of the plane.

property z_dir: *Vector*

Local Z direction normal to the plane.

class Rotation(**args*, ***kwargs*)

Subclass of Location used only for object rotation

Variables

- **X** (*float*) – rotation in degrees about X axis
- **Y** (*float*) – rotation in degrees about Y axis
- **Z** (*float*) – rotation in degrees about Z axis
- **enums,** (*optionally specify rotation ordering with Intrinsic or Extrinsic*) – defaults to Intrinsic.XYZ

class Vector(**args*, ***kwargs*)

Create a 3-dimensional vector

Parameters

- **x** (*float*) – x component
- **y** (*float*) – y component
- **z** (*float*) – z component
- **vec** (Vector | *Sequence(float)* | *gp_Vec* | *gp_Pnt* | *gp_Dir* | *gp_XYZ*) – vector representations

Note that if no z value is provided it's assumed to be zero. If no values are provided the returned Vector has the value of 0, 0, 0.

Variables
wrapped (*gp_Vec*) – the OCP vector object

property X: float

Get x value

property Y: float

Get y value

property Z: float

Get z value

__abs__() → float

Vector length operator abs()

__add__(*vec:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*) → *Vector*

Mathematical addition operator +

__copy__() → *Vector*

Return copy of self

__deepcopy__(*_memo*) → *Vector*

Return deepcopy of self

__eq__(*other: object*) → bool

Vectors equal operator ==

__mul__(*scale: float*) → *Vector*

Mathematical multiply operator *

__neg__() → *Vector*

Flip direction of vector operator -

__rmul__(*scale: float*) → *Vector*

Mathematical multiply operator *

__sub__(*vec:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*) → *Vector*

Mathematical subtraction operator -

__truediv__(*denom: float*) → *Vector*

Mathematical division operator /

add(*vec:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*) → *Vector*

Mathematical addition function

center() → *Vector*

Returns

The center of myself is myself. Provided so that vectors, vertices, and other shapes all support a common interface, when center() is requested for all objects on the stack.

cross(*vec:* Vector) → *Vector*

Mathematical cross function

distance_to_plane(*plane:* Plane) → float

Minimum unsigned distance between vector and plane

dot(*vec:* Vector) → float

Mathematical dot function

get_angle(*vec:* Vector) → float

Unsigned angle between vectors

get_signed_angle(*vec:* Vector, *normal:* Vector | *None* = *None*) → float

Signed Angle Between Vectors

Return the signed angle in degrees between two vectors with the given normal based on this math: angle = atan2((Va × Vb) Vn, Va Vb)

Parameters

- **v** (Vector) – Second Vector

- **normal** (Vector, *optional*) – normal direction. Defaults to None.

Returns
Angle between vectors

Return type
float

intersect(**args*, ***kwargs*)

Find intersection of vector and geometric object or shape

property length: float

Vector length

multiply(*scale: float*) → *Vector*

Mathematical multiply function

normalized() → *Vector*

Scale to length of 1

project_to_line(*line:* Vector) → *Vector*

Returns a new vector equal to the projection of this Vector onto the line represented by Vector <line>

Parameters
line (Vector) – project to this line

Returns
Returns the projected vector.

Return type
Vector

project_to_plane(*plane:* Plane) → *Vector*

Vector is projected onto the plane provided as input.

Parameters
args – Plane object

Returns the projected vector.
plane: Plane:

Returns:

reverse() → *Vector*

Return a vector with the same magnitude but pointing in the opposite direction

rotate(*axis:* Axis, *angle: float*) → *Vector*

Rotate about axis

Rotate about the given Axis by an angle in degrees

Parameters

- **axis** (Axis) – Axis of rotation
- **angle** (*float*) – angle in degrees

Returns
rotated vector

Return type
Vector

signed_distance_from_plane(*plane:* Plane) → float

Signed distance from plane to point vector.

sub(*vec:* Vector | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*) → *Vector*

Mathematical subtraction function

to_dir() → gp_Dir

Convert to OCCT gp_Dir object

to_pnt() → gp_Pnt

Convert to OCCT gp_Pnt object

to_tuple() → tuple[float, float, float]

Return tuple equivalent

transform(*affine_transform:* Matrix, *is_direction: bool = False*) → *Vector*

Apply affine transformation

Parameters

- **affine_transform** (*Matrix*) – affine transformation matrix
- **is_direction** (*bool, optional*) – Should self be transformed as a vector or direction? Defaults to False (vector)

Returns
transformed vector

Return type
Vector

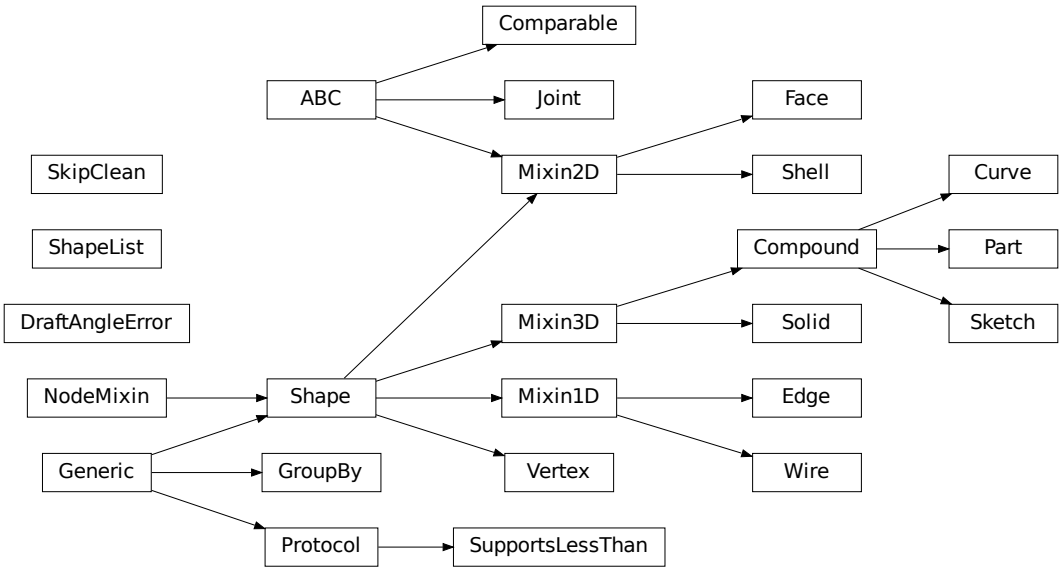
property wrapped: gp_Vec

OCCT object

1.22.2 Topological Objects

The topological object classes defined by build123d are defined below.

Note that the *Mixin1D* and *Mixin3D* classes add supplementary functionality specific to 1D (*Edge* and *Wire*) and 3D (*Compound* and ~*topology.Solid*) objects respectively. Note that a *Compound* may be contain only 1D, 2D (*Face*) or 3D objects.



class Compound(*obj: TopoDS_Compound | Iterable[Shape] | None = None*, *label: str = ''*, *color: Color | None = None*, *material: str = ''*, *joints: dict[str, Joint] | None = None*, *parent: Compound | None = None*, *children: Sequence[Shape] | None = None*)

A Compound in build123d is a topological entity representing a collection of geometric shapes grouped together within a single structure. It serves as a container for organizing diverse shapes like edges, faces, or solids. This hierarchical arrangement facilitates the construction of complex models by combining simpler shapes. Compound plays a pivotal role in managing the composition and structure of intricate 3D models in computer-aided design (CAD) applications, allowing engineers and designers to work with assemblies of shapes as unified entities for efficient modeling and analysis.

classmethod cast(*obj: TopoDS_Shape*) → *Vertex | Edge | Wire | Face | Shell | Solid | Compound*

Returns the right type of wrapper, given a OCCT object

center(*center_of: ~build123d.build_enums.CenterOf = <CenterOf.MASS>*) → Vector

Return center of object

Find center of object

Parameters
center_of (CenterOf, *optional*) – center option. Defaults to CenterOf.MASS.

Raises

- **ValueError** – Center of GEOMETRY is not supported for this object
- **NotImplementedError** – Unable to calculate center of mass of this object

Returns
center

Return type
Vector

compound() → *Compound*

Return the Compound

compounds() → *ShapeList*[*Compound*]

compounds - all the compounds in this Shape

do_children_intersect(*include_parent: bool = False*, *tolerance: float = 1e-05*) → tuple[bool, tuple[*Shape* | None, *Shape* | None], float]

Do Children Intersect

Determine if any of the child objects within a Compound/assembly intersect by intersecting each of the shapes with each other and checking for a common volume.

Parameters

- **include_parent** (*bool, optional*) – check parent for intersections. Defaults to False.
- **tolerance** (*float, optional*) – maximum allowable volume difference. Defaults to 1e-5.

Returns

do the object intersect, intersecting objects, volume of intersection

Return type

tuple[*bool*, *tuple*[Shape, Shape], *float*]

classmethod extrude(*obj:* Shell, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Compound*

Extrude a Shell into a Compound.

Parameters

direction (*VectorLike*) – direction and magnitude of extrusion

Raises

- **ValueError** – Unsupported class
- **RuntimeError** – Generated invalid result

Returns

extruded shape

Return type

Edge

get_type(*obj_type: type[*Vertex*] | type[*Edge*] | type[*Face*] | type[*Shell*] | type[*Solid*] | type[*Wire*]*) → list[*Vertex* | *Edge* | *Face* | *Shell* | *Solid* | *Wire*]

Extract the objects of the given type from a Compound. Note that this isn't the same as Faces() etc. which will extract Faces from Solids.

Parameters

obj_type (*Union[Vertex, Edge, Face, Shell, Solid, Wire]*) – Object types to extract

Returns

Extracted objects

Return type

list[*Union*[Vertex, Edge, Face, Shell, Solid, Wire]]

classmethod make_text(*txt: str*, *font_size: float*, *font: str = 'Arial'*, *font_path: ~os.PathLike[str] | str | None = None*, *font_style: ~build123d.build_enums.FontStyle = <FontStyle.REGULAR>*, *text_align: tuple[~build123d.build_enums.TextAlign, ~build123d.build_enums.TextAlign] = (<TextAlign.CENTER>, <TextAlign.CENTER>)*, *align: ~build123d.build_enums.Align | tuple[~build123d.build_enums.Align, ~build123d.build_enums.Align] | None = None*, *position_on_path: float = 0.0*, *text_path: ~topology.one_d.Edge | ~topology.one_d.Wire | None = None*, *single_line_width: float = 0.0*) → *Compound*

Text that optionally follows a path.

The text that is created can be combined as with other sketch features by specifying a mode or rotated by the given angle. In addition, edges have been previously created with arc or segment, the text will follow the path defined by these edges. The start parameter can be used to shift the text along the path to achieve precise positioning.

Parameters

- **txt** (*str*) – text to render
- **font_size** (*float*) – size of the font in model units
- **font** (*str, optional*) – font name. Defaults to "Arial"
- **font_path** (*PathLike | str, optional*) – system path to font file. Defaults to None
- **font_style** (*Font_Style, optional*) – font style, REGULAR, BOLD, BOLDITALIC, or ITALIC. Defaults to Font_Style.REGULAR
- **text_align** (*tuple[TextAlign, TextAlign], optional*) – horizontal text align LEFT, CENTER, or RIGHT. Vertical text align BOTTOM, CENTER, TOP, or TOPFIRST-LINE. Defaults to (TextAlign.CENTER, TextAlign.CENTER)
- **align** (*Align | tuple[Align, Align], optional*) – align MIN, CENTER, or MAX of object. Defaults to None
- **position_on_path** (*float, optional*) – the relative location on path to position the text, values must be between 0.0 and 1.0. Defaults to 0.0
- **text_path** – (Edge | Wire, optional): path for text to follow. Defaults to None Compound object containing multiple Shapes representing the text
- **single_line_width** (*float*) – width of outlined single line font. Defaults to 0.0

Examples:

```
fox = Compound.make_text(
    txt="The quick brown fox jumped over the lazy dog",
    font_size=10,
    position_on_path=0.1,
    text_path=jump_edge,
)
```

classmethod make_triad(*axes_scale: float*) → *Compound*

The coordinate system triad (X, Y, Z axes)

order = 4.0

project_to_viewport(*viewport_origin: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *viewport_up: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 1)*, *look_at: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *focus: float | None = None*) → tuple[*ShapeList*[*Edge*], *ShapeList*[*Edge*]]

Project a shape onto a viewport returning visible and hidden Edges.

Parameters

- **viewport_origin** (*VectorLike*) – location of viewport
- **viewport_up** (*VectorLike, optional*) – direction of the viewport y axis. Defaults to (0, 0, 1).
- **look_at** (*VectorLike, optional*) – point to look at. Defaults to None (center of shape).
- **focus** (*float, optional*) – the focal length for perspective projection Defaults to None (orthographic projection)

Returns
visible & hidden Edges

Return type
tuple[ShapeList[Edge],ShapeList[Edge]]

touch(*other:* Shape, *tolerance: float = 1e-06*) → *ShapeList*[*Vertex* | *Edge* | *Face*]

Distribute touch over compound elements.

Iterates over elements and collects touch results. Only Solid and Face elements produce boundary contacts; other shapes return empty.

Parameters

- **other** – Shape to check boundary contacts with
- **tolerance** – tolerance for contact detection

Returns
ShapeList of boundary contact geometry (empty if no contact)

unwrap(*fully: bool = True*) → Self | *Shape*

Strip unnecessary Compound wrappers

Parameters
fully (*bool, optional*) – return base shape without any Compound wrappers (otherwise one Compound is left). Defaults to True.

Returns
base shape

Return type
Union[Self, Shape]

property volume: float

volume - the volume of this Compound

class Edge(*obj: TopoDS_Edge | Axis | None | None = None*, *label: str = ''*, *color:* Color | *None = None*, *parent:* Compound | *None = None*)

An Edge in build123d is a fundamental element in the topological data structure representing a one-dimensional geometric entity within a 3D model. It encapsulates information about a curve, which could be a line, arc, or other parametrically defined shape. Edge is crucial in for precise modeling and manipulation of curves, facilitating

operations like filleting, chamfering, and Boolean operations. It serves as a building block for constructing complex structures, such as wires and faces.

property arc_center: Vector

center of an underlying circle or ellipse geometry.

close() → *Edge* | *Wire*

Close an Edge

distribute_locations(*count: int*, *start: float = 0.0*, *stop: float = 1.0*, *positions_only: bool = False*) → list[Location]

Distribute Locations

Distribute locations along edge or wire.

Parameters

- **self** – Wire:Edge:
- **count** (*int*) – Number of locations to generate
- **start** (*float*) – position along Edge|Wire to start. Defaults to 0.0.
- **stop** (*float*) – position along Edge|Wire to end. Defaults to 1.0.
- **positions_only** (*bool*) – only generate position not orientation. Defaults to False.

Returns
locations distributed along Edge|Wire

Return type
list[Location]

Raises
ValueError – count must be two or greater

classmethod extrude(*obj:* Vertex, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Edge*

Extrude a Vertex into an Edge.

Parameters
direction (*VectorLike*) – direction and magnitude of extrusion

Raises

- **ValueError** – Unsupported class
- **RuntimeError** – Generated invalid result

Returns
extruded shape

Return type
Edge

find_intersection_points(*other: Axis | Edge | None = None*, *tolerance: float = 1e-06*) → *ShapeList*[Vector]

Determine the points where a 2D edge crosses itself or another 2D edge

Parameters

- **other** (*Axis | Edge*) – curve to compare with
- **tolerance** (*float, optional*) – the precision of computing the intersection points. Defaults to TOLERANCE.

Raises
ValueError – empty edge

Returns
list of intersection points

Return type
ShapeList[Vector]

find_tangent(*angle: float*) → list[float]

Find the parameter values of self where the tangent is equal to angle.

Parameters
angle (*float*) – target angle in degrees

Returns
u values between 0.0 and 1.0

Return type
list[*float*]

geom_adaptor() → BRepAdaptor_Curve

Return the Geom Curve from this Edge

geom_equal(*other:* Edge, *tol: float = 1e-06*, *num_interpolation_points: int = 5*) → bool

Compare two edges for geometric equality within tolerance.

This compares the geometric properties of two edges, not their topological identity. Two independently created edges with the same geometry will return True.

Parameters

- **other** – Edge to compare with
- **tol** – Tolerance for numeric comparisons. Defaults to 1e-6.
- **num_interpolation_points** – Number of points to sample for unknown curve types. Defaults to 5.

Returns
True if edges are geometrically equal within tolerance

Return type
bool

property is_infinite: bool

Check if edge is infinite (LINE with length > 1e100).

classmethod make_bezier(**cntl_pnts: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *weights: list[float] | None = None*) → *Edge*

Create a rational (with weights) or non-rational bezier curve. The first and last control points represent the start and end of the curve respectively. If weights are provided, there must be one provided for each control point.

Parameters

- **cntl_pnts** (*sequence[VectorLike]*) – points defining the curve
- **weights** (*list[float], optional*) – control point weights list. Defaults to None.

Raises

- **ValueError** – Too few control points

- **ValueError** – Too many control points
- **ValueError** – A weight is required for each control point

Returns
bezier curve

Return type
Edge

classmethod make_bspline(*control_points: Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]]*, *knots: Iterable[float]*, *degree: int*, *weights: Iterable[float] | None = None*, *periodic: bool = False*) → *Edge*

Create an exact B-spline edge from control points and knot data.

Parameters

- **control_points** (*Iterable[VectorLike]*) – Control points (poles) defining the spline shape.
- **knots** (*Iterable[float]*) – Knot sequence for the spline. Repeated knot values are converted to unique knot values plus multiplicities.
- **degree** (*int*) – Polynomial degree of the spline.
- **weights** (*Iterable[float] | None, optional*) – Optional per-control-point weights for rational B-splines. Defaults to None.
- **periodic** (*bool, optional*) – Whether to create a periodic spline. Defaults to False.

Raises
ValueError – B-spline requires at least one knot.

Returns
the B-spline edge

Return type
Edge

classmethod make_circle(*radius: float*, *plane: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *start_angle: float = 360.0*, *end_angle: float = 360*, *angular_direction: ~build123d.build_enums.AngularDirection = <AngularDirection.COUNTER_CLOCKWISE>*) → *Edge*

make circle

Create a circle centered on the origin of plane

Parameters

- **radius** (*float*) – circle radius
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.
- **start_angle** (*float, optional*) – start of arc angle. Defaults to 360.0.
- **end_angle** (*float, optional*) – end of arc angle. Defaults to 360.
- **angular_direction** (*AngularDirection, optional*) – arc direction. Defaults to AngularDirection.COUNTER_CLOCKWISE.

Returns
full or partial circle

Return type
Edge

classmethod make_constrained_arcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tangency_two: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, ***, *radius: float*, *sagitta: Sagitta = Sagitta.SHORT*) → *ShapeList*[*Edge*]

classmethod make_constrained_arcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tangency_two: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, ***, *center_on: Axis | Edge*, *sagitta: Sagitta = Sagitta.SHORT*) → *ShapeList*[*Edge*]

classmethod make_constrained_arcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tangency_two: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tangency_three: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, ***, *sagitta: Sagitta = Sagitta.SHORT*) → *ShapeList*[*Edge*]

classmethod make_constrained_arcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, ***, *center: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *ShapeList*[*Edge*]

classmethod make_constrained_arcs(*tangency_one: tuple[Axis | Edge, Tangency] | Axis | Edge | Vertex | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, ***, *radius: float*, *center_on: Edge*) → *ShapeList*[*Edge*]

classmethod make_constrained_lines(*tangency_one: tuple[Edge, Tangency] | Axis | Edge*, *tangency_two: tuple[Edge, Tangency] | Axis | Edge*) → *ShapeList*[*Edge*]

classmethod make_constrained_lines(*tangency_one: tuple[Edge, Tangency] | Edge*, *tangency_two: Vector*) → *ShapeList*[*Edge*]

classmethod make_constrained_lines(*tangency_one: tuple[Edge, Tangency] | Edge*, *tangency_two: Axis*, ***, *angle: float | None = None*, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → *ShapeList*[*Edge*]

Create planar line(s) on XY subject to tangency/contact constraints.

Supported cases

1. Tangent to two curves
2. Tangent to one curve and passing through a given point

classmethod make_ellipse(*x_radius: float*, *y_radius: float*, *plane: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *start_angle: float = 360.0*, *end_angle: float = 360.0*, *angular_direction: ~build123d.build_enums.AngularDirection = <AngularDirection.COUNTER_CLOCKWISE>*) → *Edge*

make ellipse

Makes an ellipse centered at the origin of plane.

Parameters

- **x_radius** (*float*) – x radius of the ellipse (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the ellipse (along the y-axis of plane)
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.
- **start_angle** (*float, optional*) – Defaults to 360.0.
- **end_angle** (*float, optional*) – Defaults to 360.0.
- **angular_direction** (*AngularDirection, optional*) – arc direction. Defaults to AngularDirection.COUNTER_CLOCKWISE.

Returns
full or partial ellipse

Return type
Edge

classmethod make_helix(*pitch: float*, *height: float*, *radius: float*, *center: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 0)*, *normal: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 1)*, *angle: float = 0.0*, *lefthand: bool = False*) → *Wire*

Make a helix with a given pitch, height and radius. By default a cylindrical surface is used to create the helix. If the :angle: is set (the apex given in degree) a conical surface is used instead.

Parameters

- **pitch** (*float*) – distance per revolution along normal
- **height** (*float*) – total height
- **radius** (*float*)
- **center** (*VectorLike, optional*) – Defaults to (0, 0, 0).
- **normal** (*VectorLike, optional*) – Defaults to (0, 0, 1).
- **angle** (*float, optional*) – conical angle. Defaults to 0.0.
- **lefthand** (*bool, optional*) – Defaults to False.

Returns
helix

Return type
Wire

classmethod make_hyperbola(*x_radius: float*, *y_radius: float*, *plane: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *start_angle: float = 360.0*, *end_angle: float = 360.0*, *angular_direction: ~build123d.build_enums.AngularDirection = <AngularDirection.COUNTER_CLOCKWISE>*) → *Edge*

make hyperbola

Makes a hyperbola centered at the origin of plane.

Parameters

- **x_radius** (*float*) – x radius of the hyperbola (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the hyperbola (along the y-axis of plane)
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.

- **start_angle** (*float, optional*) – Defaults to 360.0.
- **end_angle** (*float, optional*) – Defaults to 360.0.
- **angular_direction** (*AngularDirection, optional*) – arc direction. Defaults to AngularDirection.COUNTER_CLOCKWISE.

Returns
full or partial hyperbola

Return type
Edge

classmethod make_line(*point1: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *point2: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Edge*

Create a line between two points

Parameters

- **point1** – VectorLike: that represents the first point
- **point2** – VectorLike: that represents the second point

Returns
A linear edge between the two provided points

classmethod make_mid_way(*first:* Edge, *second:* Edge, *middle: float = 0.5*) → *Edge*

make line between edges

Create a new linear Edge between the two provided Edges. If the Edges are parallel but in the opposite directions one Edge is flipped such that the mid way Edge isn't truncated.

Parameters

- **first** (Edge) – first reference Edge
- **second** (Edge) – second reference Edge
- **middle** (*float, optional*) – factional distance between Edges. Defaults to 0.5.

Returns
linear Edge between two Edges

Return type
Edge

classmethod make_parabola(*focal_length: float*, *plane: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *start_angle: float = 0.0*, *end_angle: float = 90.0*, *angular_direction: ~build123d.build_enums.AngularDirection = <AngularDirection.COUNTER_CLOCKWISE>*) → *Edge*

make parabola

Makes an parabola centered at the origin of plane.

Parameters

- **focal_length** (*float*) – focal length the parabola (distance from the vertex to focus along the x-axis of plane)
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.
- **start_angle** (*float, optional*) – Defaults to 0.0.
- **end_angle** (*float, optional*) – Defaults to 90.0.

- **angular_direction** (*AngularDirection, optional*) – arc direction. Defaults to AngularDirection.COUNTER_CLOCKWISE.

Returns
full or partial parabola

Return type
Edge

classmethod make_spline(*points: list[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]]*, *tangents: list[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]] | None = None*, *periodic: bool = False*, *parameters: list[float] | None = None*, *scale: bool = True*, *tol: float = 1e-06*) → Edge

Spline

Interpolate a spline through the provided points.

Parameters

- **points** (*list[VectorLike]*) – the points defining the spline
- **tangents** (*list[VectorLike], optional*) – start and finish tangent. Defaults to None.
- **periodic** (*bool, optional*) – creation of periodic curves. Defaults to False.
- **parameters** (*list[float], optional*) – the value of the parameter at each interpolation point. (The interpolated curve is represented as a vector-valued function of a scalar parameter.) If periodic == True, then len(parameters) must be len(interpolation points) + 1, otherwise len(parameters) must be equal to len(interpolation points). Defaults to None.
- **scale** (*bool, optional*) – whether to scale the specified tangent vectors before interpolating. Each tangent is scaled, so it's length is equal to the derivative of the Lagrange interpolated curve. I.e., set this to True, if you want to use only the direction of the tangent vectors specified by *tangents* , but not their magnitude. Defaults to True.
- **tol** (*float, optional*) – tolerance of the algorithm (consult OCC documentation). Used to check that the specified points are not too close to each other, and that tangent vectors are not too short. (In either case interpolation may fail.). Defaults to 1e-6.

Raises

- **ValueError** – Parameter for each interpolation point
- **ValueError** – Tangent for each interpolation point
- **ValueError** – B-spline interpolation failed

Returns
the spline

Return type
Edge

classmethod make_spline_approx(*points: list[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]]*, *tol: float = 0.001*, *smoothing: tuple[float, float, float] | None = None*, *min_deg: int = 1*, *max_deg: int = 6*) → Edge

Approximate a spline through the provided points.

Parameters

- **points** (*list[Vector]*)
- **tol** (*float, optional*) – tolerance of the algorithm. Defaults to 1e-3.

- **smoothing** (*Tuple[float, float, float], optional*) – optional tuple of 3 weights use for variational smoothing. Defaults to None.
- **min_deg** (*int, optional*) – minimum spline degree. Enforced only when smoothing is None. Defaults to 1.
- **max_deg** (*int, optional*) – maximum spline degree. Defaults to 6.

Raises
ValueError – B-spline approximation failed

Returns
spline

Return type
Edge

classmethod make_tangent_arc(*start: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tangent: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *end: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Edge*

Tangent Arc

Makes a tangent arc from point start, in the direction of tangent and ends at end.

Parameters

- **start** (*VectorLike*) – start point
- **tangent** (*VectorLike*) – start tangent
- **end** (*VectorLike*) – end point

Returns
circular arc

Return type
Edge

classmethod make_three_point_arc(*point1: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *point2: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *point3: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Edge*

Three Point Arc

Makes a three point arc through the provided points

Parameters

- **point1** (*VectorLike*) – start point
- **point2** (*VectorLike*) – middle point
- **point3** (*VectorLike*) – end point

Returns
a circular arc through the three points

Return type
Edge

order = 1.0

param_at(*position: float*) → float

Map a normalized arc-length position to the underlying OCCT parameter.

Returns the native OCCT curve parameter corresponding to the given normalized *position* (0.0 → start, 1.0 → end). For closed/periodic edges, OCCT may return a value **outside** the edge's nominal parameter range *[param_min, param_max]* (e.g., by adding/subtracting multiples of the period). If you require a value folded into the edge's range, apply a modulo with the parameter span.

Parameters
position (*float*) – Normalized arc-length position along the shape, where *0.0* is the start and *1.0* is the end. Values outside *[0.0, 1.0]* are not validated and yield OCCT-dependent results.

Returns
OCCT parameter (for edges) **or** composite "edgeIndex + fraction" parameter (for wires), as described above.

Return type
float

param_at_point(*point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → float

Return the normalized parameter ([0.0, 1.0]) of the location on this edge closest to *point*.

This method always returns a **normalized** parameter across the edge's full OCCT parameter range, even though the underlying OCP/OCCT queries work in native (non-normalized) parameters. It is robust to several OCCT quirks:

1) Vertex snap (fast path) If *point* coincides (within tolerance) with one of the edge's vertices, that vertex's OCCT parameter is used and normalized to [0, 1]. Note: for a closed edge, a vertex may represent both start and end; the mapping is therefore ambiguous and either end may be chosen.

2) Projection via GeomAPI_ProjectPointOnCurve The OCCT projector's *LowerDistanceParameter()* can legitimately return a value **outside** the edge's [param_min, param_max] (e.g., periodic curves or implementation behavior). The result is wrapped back into range using a modulo by the parameter span and then normalized to [0, 1]. The projected answer is accepted only if re-evaluating the 3D point at that normalized parameter is within tolerance of the input *point*.

3) Fallback numeric search (robust path) If the projector fails the validation, a bounded 1D search is performed over [0, 1] using progressive subdivision and local minimization of the 3D distance ‖edge(u) - point‖. The first minimum found under geometric resolution is returned.

Parameters
point (*VectorLike*) – A point expected to lie on this edge (within tolerance).

Raises

- **ValueError** – If *point* is not on the edge within tolerance.
- **ValueError** – Can't find param on empty edge
- **RuntimeError** – If no parameter can be found (e.g., extremely pathological curves or numerical failure).

Returns
Normalized parameter in [0.0, 1.0] corresponding to the point's closest location on the edge.

Return type
float

project_to_shape(*target_object:* Shape, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *center: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → *ShapeList*[*Edge*]

Project Edge

Project an Edge onto a Shape generating new wires on the surfaces of the object one and only one of *direction* or *center* must be provided. Note that one or more wires may be generated depending on the topology of the target object and location/direction of projection.

To avoid flipping the normal of a face built with the projected wire the orientation of the output wires are forced to be the same as self.

Parameters

- **target_object** – Object to project onto
- **direction** – Parallel projection direction. Defaults to None.
- **center** – Conical center of projection. Defaults to None.
- **target_object** – Shape:
- **direction** – VectorLike: (Default value = None)
- **center** – VectorLike: (Default value = None)

Returns
Projected Edge(s)

Raises
ValueError – Only one of direction or center must be provided

reversed(*reconstruct: bool = False*) → *Edge*

Return a copy of self with the opposite orientation.

Parameters
reconstruct (*bool, optional*) – rebuild edge instead of setting OCCT flag. Defaults to False.

Returns
reversed

Return type
Edge

to_axis() → Axis

Translate a linear Edge to an Axis

to_wire() → *Wire*

Edge as Wire

trim(*start: float | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *end: float | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Edge*

Create a new edge by keeping only the section between start and end.

Parameters

- **start** (*float | VectorLike*) – 0.0 <= start < 1.0 or point on edge
- **end** (*float | VectorLike*) – 0.0 < end <= 1.0 or point on edge

Raises

- **TypeError** – invalid input, must be float or VectorLike
- **ValueError** – can't trim empty edge

Returns
trimmed edge

Return type
Edge

trim_infinite(*half_length: float*) → *Edge*

Trim an infinite line edge to a finite length.

OCCT's boolean operations struggle with very long edges (length > 1e100). This method trims such edges to a reasonable size centered at edge.center().

For non-infinite edges, returns self unchanged.

Parameters
half_length – Half-length of the resulting edge

Returns
Trimmed edge if infinite, otherwise self

trim_to_length(*start: float | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *length: float*) → *Edge*

Create a new edge starting at the given normalized parameter of a given length.

Parameters

- **start** (*float | VectorLike*) – 0.0 <= start < 1.0 or point on edge
- **length** (*float*) – target length

Raises
ValueError – can't trim empty edge

Returns
trimmed edge

Return type
Edge

trim_to_other(*other:* Shape *| Axis | Location | Plane | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Edge* | None

Return the shortest Edge of self trimmed by other or None if they don't intersect

class Face(*obj: TopoDS_Face | Plane*, *label: str = ''*, *color: Color | None = None*, *parent:* Compound *| None = None*)

class Face(*outer_wire:* Wire, *inner_wires: Iterable[*Wire*] | None = None*, *label: str = ''*, *color: Color | None = None*, *parent:* Compound *| None = None*)

A Face in build123d represents a 3D bounded surface within the topological data structure. It encapsulates geometric information, defining a face of a 3D shape. These faces are integral components of complex structures, such as solids and shells. Face enables precise modeling and manipulation of surfaces, supporting operations like trimming, filleting, and Boolean operations.

property area_without_holes: float

Calculate the total surface area of the face, including the areas of any holes.

This property returns the overall area of the face as if the inner boundaries (holes) were filled in.

Returns
The total surface area, including the area of holes. Returns 0.0 if the face is empty.

Return type
float

property axes_of_symmetry: list[Axis]

Computes and returns the axes of symmetry for a planar face.

The method determines potential symmetry axes by analyzing the face's geometry:

- It first validates that the face is non-empty and planar.
- For faces with inner wires (holes), it computes the centroid of the holes and the face's overall center (COG).
  - If the holes' centroid significantly deviates from the COG (beyond a specified tolerance), the symmetry axis is taken along the line connecting these points; otherwise, each hole's center is used to generate a candidate axis.
- For faces without holes, candidate directions are derived by sampling midpoints along the outer wire's edges.
  - If curved edges are present, additional candidate directions are obtained from an oriented bounding box (OBB) constructed around the face.

For each candidate direction, the face is split by a plane (defined using the candidate direction and the face's normal). The top half of the face is then mirrored across this plane, and if the area of the intersection between the mirrored half and the bottom half matches the bottom half's area within a small tolerance, the direction is accepted as an axis of symmetry.

Returns

A list of Axis objects, each defined by the face's
center and a direction vector, representing the symmetry axes of the face.

Return type
list[Axis]

Raises

- **ValueError** – If the face or its underlying representation is empty.
- **ValueError** – If the face is not planar.

property axis_of_rotation: None | Axis

Get the rotational axis of a cylinder or torus

center(*center_of: ~build123d.build_enums.CenterOf = <CenterOf.GEOMETRY>*) → Vector

Center of Face

Return the center based on center_of

Parameters
center_of (CenterOf, *optional*) – centering option. Defaults to CenterOf.GEOMETRY.

Returns
center

Return type
Vector

property center_location: Location

Location at the center of face

chamfer_2d(*distance: float*, *distance2: float*, *vertices: Iterable[*Vertex*]*, *edge:* Edge | *None = None*) → *Face*

Apply 2D chamfer to a face

Parameters

- **distance** (*float*) – chamfer length
- **distance2** (*float*) – chamfer length
- **vertices** (*Iterable[Vertex]*) – vertices to chamfer
- **edge** (Edge) – identifies the side where length is measured. The vertices must be part of the edge

Raises

- **ValueError** – Cannot chamfer at this location
- **ValueError** – One or more vertices are not part of edge

Returns
face with a chamfered corner(s)

Return type
Face

classmethod extrude(*obj:* Edge, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Face*

Extrude an Edge into a Face.

Parameters
direction (*VectorLike*) – direction and magnitude of extrusion

Raises

- **ValueError** – Unsupported class
- **RuntimeError** – Generated invalid result

Returns
extruded shape

Return type
Face

fillet_2d(*radius: float*, *vertices: Iterable[*Vertex*]*) → *Face*

Apply 2D fillet to a face

Parameters

- **radius** – float:
- **vertices** – Iterable[Vertex]:

Returns:

geom_adaptor() → Geom_Surface

Return the Geom Surface for this Face

property geometry: None | str

geometry of planar face

inner_wires() → *ShapeList*[*Wire*]

Extract the inner or hole wires from this Face

property is_circular_concave: bool

Determine whether a given face is concave relative to its underlying geometry for supported geometries: cylinder, sphere, torus.

Returns
True if concave; otherwise, False.

Return type
bool

property is_circular_convex: bool

Determine whether a given face is convex relative to its underlying geometry for supported geometries: cylinder, sphere, torus.

Returns
True if convex; otherwise, False.

Return type
bool

is_coplanar(*plane: Plane*) → bool

Is this planar face coplanar with the provided plane

is_inside(*point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tolerance: float = 1e-06*) → bool

Point inside Face

Returns whether or not the point is inside a Face within the specified tolerance. Points on the edge of the Face are considered inside.

Parameters

- **point** (*VectorLike*) – tuple or Vector representing 3D point to be tested
- **tolerance** (*float*) – tolerance for inside determination. Defaults to 1.0e-6.
- **point** – VectorLike:
- **tolerance** – float: (Default value = 1.0e-6)

Returns
indicating whether or not point is within Face

Return type
bool

property is_planar: Plane | None

Is the face planar even though its geom_type may not be PLANE - if so return Plane

property length: None | float

length of planar face

location_at(*surface_point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, ***, *x_dir: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → Location

location_at(*u: float*, *v: float*, ***, *x_dir: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → Location

location_at

Get the location (origin and orientation) on the surface of the face.

This method supports two overloads:

1. *location_at(u: float, v: float, *, x_dir: VectorLike | None = None) -> Location* - Specifies the point in normalized UV parameter space of the face. - *u* and *v* are floats between 0.0 and 1.0. - Optionally override the local X direction using *x_dir*.

2. *location_at(surface_point: VectorLike, *, x_dir: VectorLike | None = None) -> Location* - Projects the given 3D point onto the face surface. - The point must be reasonably close to the face. - Optionally override the local X direction using *x_dir*.

If no arguments are provided, the location at the center of the face (u=0.5, v=0.5) is returned.

Parameters

- **u** (*float*) – Normalized horizontal surface parameter (optional).
- **v** (*float*) – Normalized vertical surface parameter (optional).
- **surface_point** (*VectorLike*) – A 3D point near the surface (optional).
- **x_dir** (*VectorLike, optional*) – Direction for the local X axis. If not given, the tangent in the U direction is used.

Returns
A full 3D placement at the specified point on the face surface.

Return type
Location

Raises
ValueError – If only one of *u* or *v* is provided or invalid keyword args are passed.

classmethod make_bezier_surface(*points: list[list[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]]]*, *weights: list[list[float]] | None = None*) → *Face*

Construct a Bézier surface from the provided 2d array of points.

Parameters

- **points** (*list[list[VectorLike]]*) – a 2D list of control points
- **weights** (*list[list[float]], optional*) – control point weights. Defaults to None.

Raises

- **ValueError** – Too few control points
- **ValueError** – Too many control points
- **ValueError** – A weight is required for each control point

Returns
a potentially non-planar face

Return type
Face

classmethod make_gordon_surface(*profiles: Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | Edge]*, *guides: Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | Edge]*, *tolerance: float = 0.0003*) → *Face*

Constructs a Gordon surface from a network of profile and guide curves.

Requirements: 1. Profiles and guides may be defined as points or curves. 2. Only the first or last profile or guide may be a point. 3. At least one profile and one guide must be a non-point curve. 4. Each profile must intersect with every guide. 5. Both ends of every profile must lie on a guide. 6. Both ends of every guide must lie on a profile.

Parameters

- **profiles** (*Iterable[VectorLike | Edge]*) – Profiles defined as points or edges.

- **guides** (*Iterable[VectorLike | Edge]*) – Guides defined as points or edges.
- **tolerance** (*float, optional*) – Tolerance used for surface construction and intersection calculations.

Raises
ValueError – input Edge cannot be empty.

Returns
the interpolated Gordon surface

Return type
Face

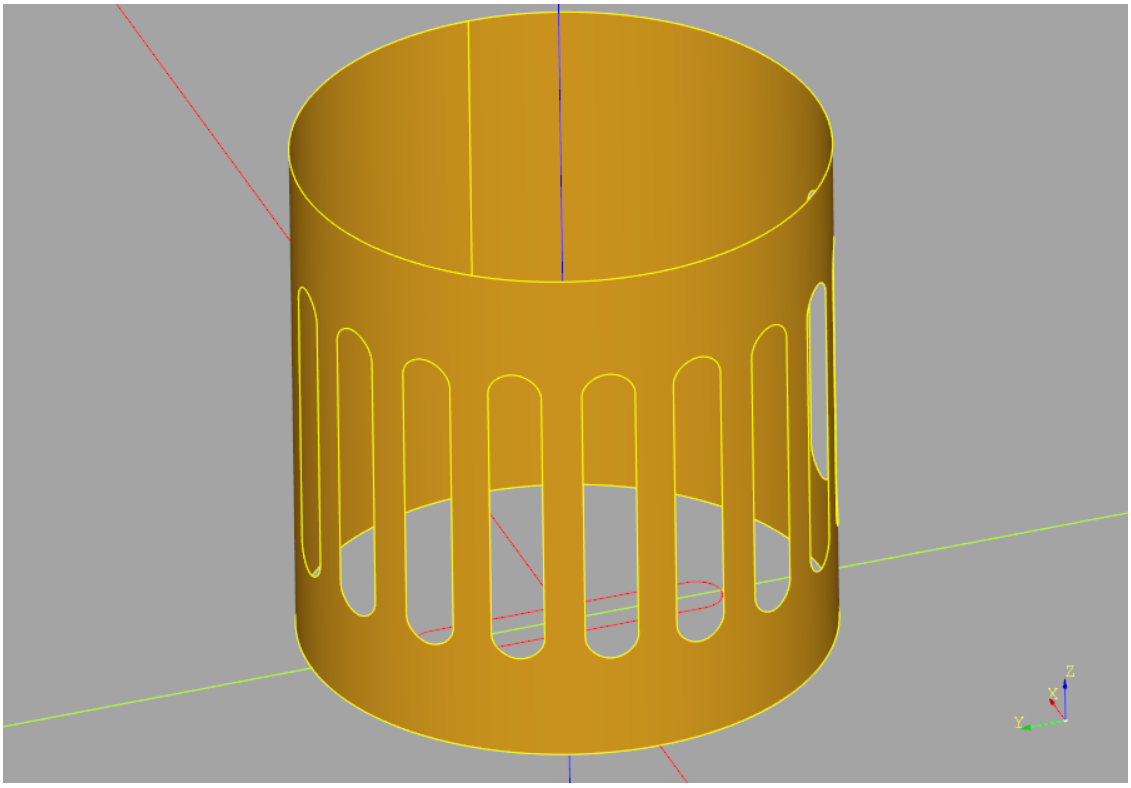
make_holes(*interior_wires: list[Wire]*) → *Face*

Make Holes in Face

Create holes in the Face 'self' from interior_wires which must be entirely interior. Note that making holes in faces is more efficient than using boolean operations with solid object. Also note that OCCT core may fail unless the orientation of the wire is correct - use *Wire(forward_wire.wrapped.Reversed())* to reverse a wire.

Example

For example, make a series of slots on the curved walls of a cylinder.



Parameters

- **interior_wires** – a list of hole outline wires
- **interior_wires** – list[Wire]:

Returns
'self' with holes

Return type
Face

Raises

- **RuntimeError** – adding interior hole in non-planar face with provided interior_wires
- **RuntimeError** – resulting face is not valid

classmethod make_plane(*plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Face*

Create a unlimited size Face aligned with plane

classmethod make_rect(*width: float*, *height: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Face*

Make a Rectangle centered on center with the given normal

Parameters

- **width** (*float, optional*) – width (local x).
- **height** (*float, optional*) – height (local y).
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.

Returns
The centered rectangle

Return type
Face

classmethod make_surface(*exterior: Wire | Iterable[Edge]*, *surface_points: Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]] | None = None*, *interior_wires: Iterable[Wire] | None = None*) → *Face*

Create Non-Planar Face

Create a potentially non-planar face bounded by exterior (wire or edges), optionally refined by surface_points with optional holes defined by interior_wires.

Parameters

- **exterior** (*Union[Wire, list[Edge]]*) – Perimeter of face
- **surface_points** (*list[VectorLike], optional*) – Points on the surface that refine the shape. Defaults to None.
- **interior_wires** (*list[Wire], optional*) – Hole(s) in the face. Defaults to None.

Raises

- **RuntimeError** – Internal error building face
- **RuntimeError** – Error building non-planar face with provided surface_points
- **RuntimeError** – Error adding interior hole
- **RuntimeError** – Generated face is invalid

Returns
Potentially non-planar face

Return type
Face

classmethod make_surface_from_array_of_points(*points: list[list[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]]]*, *tol: float = 0.01*, *smoothing: tuple[float, float, float] | None = None*, *min_deg: int = 1*, *max_deg: int = 3*) → *Face*

Approximate a spline surface through the provided 2d array of points. The first dimension correspond to points on the vertical direction in the parameter space of the face. The second dimension correspond to points on the horizontal direction in the parameter space of the face. The 2 dimensions are U,V dimensions of the parameter space of the face.

Parameters

- **points** (*list[list[VectorLike]]*) – a 2D list of points, first dimension is V parameters second is U parameters.
- **tol** (*float, optional*) – tolerance of the algorithm. Defaults to 1e-2.
- **smoothing** (*Tuple[float, float, float], optional*) – optional tuple of 3 weights use for variational smoothing. Defaults to None.
- **min_deg** (*int, optional*) – minimum spline degree. Enforced only when smoothing is None. Defaults to 1.
- **max_deg** (*int, optional*) – maximum spline degree. Defaults to 3.

Raises

ValueError – B-spline approximation failed

Returns

a potentially non-planar face defined by points

Return type

Face

classmethod make_surface_from_curves(*edge1:* Edge, *edge2:* Edge) → *Face*

classmethod make_surface_from_curves(*wire1:* Wire, *wire2:* Wire) → *Face*

make_surface_from_curves

Create a ruled surface out of two edges or two wires. If wires are used then these must have the same number of edges.

Parameters

- **curve1** (*Union[Edge,Wire]*) – side of surface
- **curve2** (*Union[Edge,Wire]*) – opposite side of surface

Returns

potentially non planar surface

Return type

Face

classmethod make_surface_patch(*edge_face_constraints: Iterable[tuple[*Edge, Face, *ContinuityLevel]] | None = None*, *edge_constraints: Iterable[*Edge*] | None = None*, *point_constraints: Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]] | None = None*) → *Face*

Create a potentially non-planar face patch bounded by exterior edges which can be optionally refined using support faces to ensure e.g. tangent surface continuity. Also can optionally refine the surface using surface points.

Parameters

- **edge_face_constraints** (*list[tuple[Edge, Face, ContinuityLevel]], optional*) – Edges defining perimeter of face with adjacent support faces subject to ContinuityLevel. Defaults to None.
- **edge_constraints** (*list[Edge], optional*) – Edges defining perimeter of face without adjacent support faces. Defaults to None.
- **point_constraints** (*list[VectorLike], optional*) – Points on the surface that refine the shape. Defaults to None.

Raises

- **RuntimeError** – Error building non-planar face with provided constraints
- **RuntimeError** – Generated face is invalid

Returns
Potentially non-planar face

Return type
Face

normal_at(*surface_point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → Vector

normal_at(*u: float*, *v: float*) → Vector

normal_at

Computes the normal vector at the desired location on the face.

Parameters
surface_point (*VectorLike, optional*) – a point that lies on the surface where the normal. Defaults to None.

Returns
surface normal direction

Return type
Vector

order = 2.0

outer_wire() → *Wire*

Extract the perimeter wire from this Face

position_at(*u: float*, *v: float*) → Vector

Computes a point on the Face given u, v coordinates.

Parameters

- **u** (*float*) – the horizontal coordinate in the parameter space of the Face, between 0.0 and 1.0
- **v** (*float*) – the vertical coordinate in the parameter space of the Face, between 0.0 and 1.0

Returns
point on Face

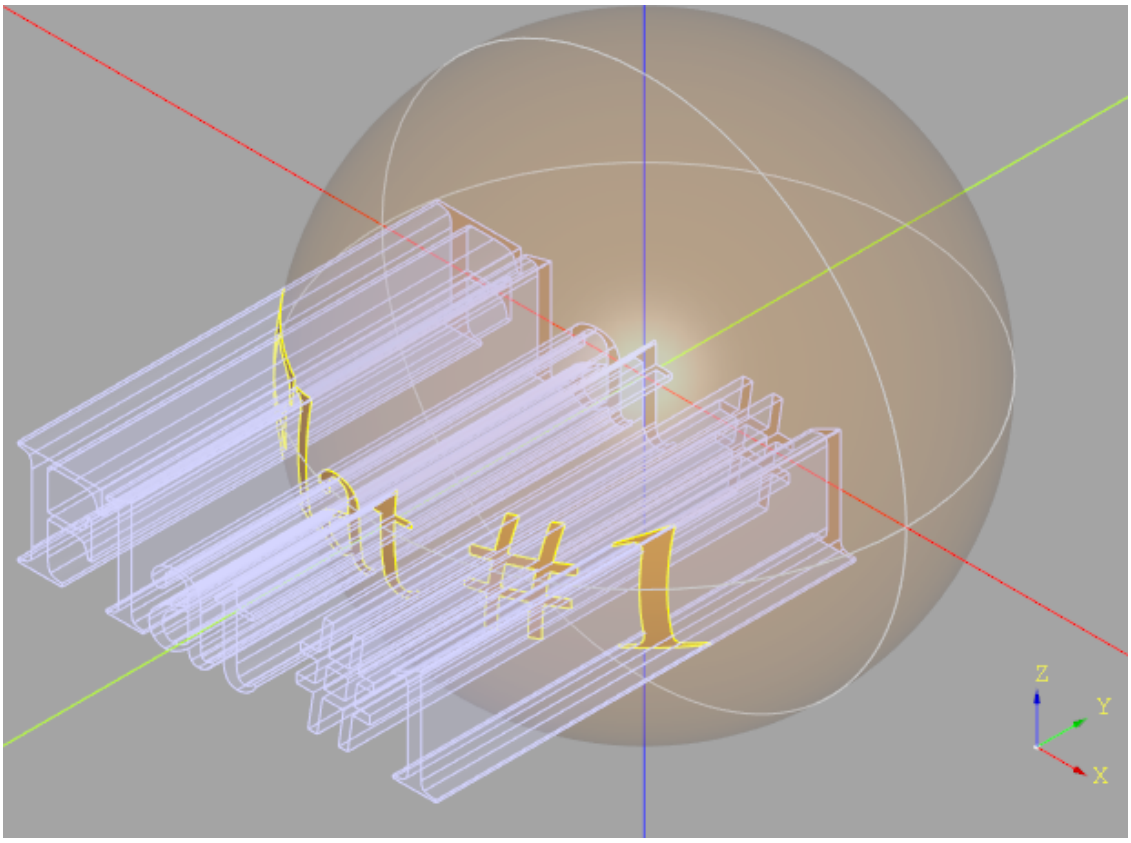
Return type
Vector

project_to_shape(*target_object:* Shape, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *ShapeList*[*Face* | *Shell*]

Project Face to target Object

Project a Face onto a Shape generating new Face(s) on the surfaces of the object.

A projection with no taper is illustrated below:



Note that an array of faces is returned as the projection might result in faces on the "front" and "back" of the object (or even more if there are intermediate surfaces in the projection path). faces "behind" the projection are not returned.

Parameters

- **target_object** (Shape) – Object to project onto
- **direction** (*VectorLike*) – projection direction

Returns

Face(s) projected on target object ordered by distance

Return type

ShapeList[Face]

property radii: None | tuple[float, float]

Return the major and minor radii of a torus otherwise None

property radius: None | float

Return the radius of a cylinder or sphere, otherwise None

classmethod revolve(*profile:* Edge, *angle: float*, *axis: Axis*) → *Face*

sweep

Revolve an Edge around an axis.

Parameters

- **profile** (Edge) – the object to sweep
- **angle** (*float*) – the angle to revolve through
- **axis** (Axis) – rotation Axis

Returns
resulting face

Return type
Face

property seams: *ShapeList*[*Edge*]

Return the seams contained within this Face

property semi_angle: None | float

Return the semi angle of a cone, otherwise None

classmethod sew_faces(*faces: Iterable[*Face*]*) → list[*ShapeList*[*Face*]]

sew faces

Group contiguous faces and return them in a list of ShapeList

Parameters
faces (*Iterable[*Face*]*) – Faces to sew together

Raises
RuntimeError – OCCT SewedShape generated unexpected output

Returns
grouped contiguous faces

Return type
list[ShapeList[Face]]

classmethod sweep(*profile: Curve | Edge | Wire*, *path: Curve | Edge | Wire*, *transition=<Transition.TRANSFORMED>*) → *Face*

Sweep a 1D profile along a 1D path. Both the profile and path must be composed of only 1 Edge.

Parameters

- **profile** (*Union[*Curve*,*Edge*,*Wire*]*) – the object to sweep
- **path** (*Union[*Curve*,*Edge*,*Wire*]*) – the path to follow when sweeping
- **transition** (Transition, *optional*) – handling of profile orientation at C1 path discontinuities. Defaults to Transition.TRANSFORMED.

Raises
ValueError – Only 1 Edge allowed in profile & path

Returns
resulting face, may be non-planar

Return type
Face

to_arcs(*tolerance: float = 0.001*) → *Face*

Approximate planar face with arcs and straight line segments.

This is a utility used internally to convert or adapt a face for Boolean operations. Its purpose is not typically for general use, but rather as a helper within the Boolean kernel to ensure input faces are in a compatible and canonical form.

Parameters
tolerance (*float, optional*) – Approximation tolerance. Defaults to 1e-3.

Returns
approximated face

Return type
Face

property uv_face: *Face*

Create a planar face from a face's parametric-space boundary.

Each boundary edge's pcurve on `self` is converted to a normal build123d `Edge` on the XY plane, where X is the surface U parameter and Y is the surface V parameter. The original outer/inner wire structure is kept so the result can be displayed with normal build123d/ocp-vscode tooling.

Parameters
source_face – Planar or non-planar face to inspect.

Returns
A planar `Face` in UV parameter space.

property volume: float

volume - the volume of this Face, which is always zero

property width: None | float

width of planar face

wire() → *Wire*

Return the outerwire, generate a warning if inner_wires present

without_holes() → *Face*

Remove all of the holes from this face.

Returns
A new Face instance identical to the original but without any holes.

Return type
Face

wrap(*planar_shape:* Edge, *surface_loc: Location*, *tolerance: float = 0.001*, *extension_factor: float = 0.1*) → *Edge*

wrap(*planar_shape:* Wire, *surface_loc: Location*, *tolerance: float = 0.001*, *extension_factor: float = 0.1*) → *Wire*

wrap(*planar_shape:* Face, *surface_loc: Location*, *tolerance: float = 0.001*, *extension_factor: float = 0.1*) → *Face*

wrap

Wrap a planar 2D shape onto a 3D surface.

This method conforms a 2D shape defined on the XY plane (Edge, Wire, or Face) to the curvature of a non-planar 3D Face (the target surface), starting at a specified surface location. The operation attempts

to preserve the original edge lengths and shape as closely as possible while minimizing the geometric distortion that naturally arises when mapping flat geometry onto curved surfaces.

The wrapping process follows the local orientation of the surface and progressively fits each edge along the curvature. To help ensure continuity, the first and last edges are extended and trimmed to close small gaps introduced by distortion. The final shape is tightly aligned to the surface geometry.

This method is useful for applying flat features—such as decorative patterns, cutouts, or boundary outlines—onto curved or freeform surfaces while retaining their original proportions.

Parameters

- **planar_shape** (Edge | Wire | Face) – flat shape to wrap around surface
- **surface_loc** (Location) – location on surface to wrap
- **tolerance** (*float, optional*) – maximum allowed error. Defaults to 0.001
- **extension_factor** (*float, optional*) – amount to extend the wrapped first and last edges to allow them to cross. Defaults to 0.1

Raises

ValueError – Invalid planar shape

Returns

wrapped shape

Return type

Edge | Wire | Face

wrap_faces(*faces: Iterable[*Face*]*, *path:* Wire | Edge, *start: float = 0.0*) → *ShapeList*[*Face*]

Wrap a sequence of 2D faces onto a 3D surface, aligned along a guiding path.

This method places multiple planar *Face* objects (defined in the XY plane) onto a curved 3D surface (*self*), following a given path (Wire or Edge) that lies on or closely follows the surface. Each face is spaced along the path according to its original horizontal (X-axis) position, preserving the relative layout of the input faces.

The wrapping process attempts to maintain the shape and size of each face while minimizing distortion. Each face is repositioned to the origin, then individually wrapped onto the surface starting at a specific point along the path. The face's new orientation is defined using the path's tangent direction and the surface normal at that point.

This is particularly useful for placing a series of features—such as embossed logos, engraved labels, or patterned tiles—onto a freeform or cylindrical surface, aligned along a reference edge or curve.

Parameters

- **faces** (*Iterable[*Face*]*) – An iterable of 2D planar faces to be wrapped.
- **path** (Wire | Edge) – A curve on the target surface that defines the alignment direction. The X-position of each face is mapped to a relative position along this path.
- **start** (*float, optional*) – The relative starting point on the path (between 0.0 and 1.0) where the first face should be placed. Defaults to 0.0.

Returns

A list of wrapped face objects, aligned and conformed to the surface.

Return type

ShapeList[Face]

class Mixin1D(*obj: TopoDS_Shape | None = None*, *label: str = ''*, *color: ColorLike | None = None*, *parent: Compound | None = None*)

Methods to add to the Edge and Wire classes

__matmul__(*position: float*) → Vector

Position on wire operator @

__mod__(*position: float*) → Vector

Tangent on wire operator %

classmethod cast(*obj: TopoDS_Shape*) → *Vertex* | *Edge* | *Wire*

Returns the right type of wrapper, given a OCCT object

center(*center_of: ~build123d.build_enums.CenterOf = <CenterOf.GEOMETRY>*) → Vector

Center of object

Return the center based on center_of

Parameters
center_of (*CenterOf, optional*) – centering option. Defaults to CenterOf.GEOMETRY.

Returns
center

Return type
Vector

common_plane(**lines: Edge | Wire | None*, *tolerance: float = 1e-06*) → None | Plane

Find the plane containing all the edges/wires (including self). If there is no common plane return None. If the edges are coaxial, select one of the infinite number of valid planes.

Parameters

- **lines** (*sequence of Edge | Wire*) – edges in common with self
- **tolerance** (*float*) – amount lines can deviate from plane. Defaults to TOLERANCE.

Returns
Either the common plane or None

Return type
None | Plane

curvature_comb(*count: int = 100*, *max_tooth_size: float | None = None*) → *ShapeList*[*Edge*]

Build a *curvature comb* for a planar (XY) 1D curve.

A curvature comb is a set of short line segments ("teeth") erected perpendicular to the curve that visualize the signed curvature (u). Tooth length is proportional to ||κ|| and the direction encodes the sign (left normal for κ>0, right normal for κ<0). This is useful for inspecting fairness and continuity (C0/C1/C2) of edges and wires.

Parameters

- **count** (*int, optional*) – Number of uniformly spaced samples over the normalized parameter. Increase for a denser comb. Defaults to 100.
- **max_tooth_size** (*float | None, optional*) – Maximum tooth height in model units. If None, set to 10% maximum curve dimension. Defaults to None.

Raises

- **ValueError** – Empty curve.

- **ValueError** – If the curve is not planar on *Plane.XY*.

Returns
A list of short *Edge* objects (lines) anchored on the curve and oriented along the left normal $n = normalize(t) \times +Z$.

Return type
ShapeList[Edge]

Notes

- On circles, $\kappa = 1/R$ so tooth length is constant.
- On straight segments, $\kappa = 0$ so no teeth are drawn.
- At inflection points $\kappa \to 0$ and the tooth flips direction.
- At C0 corners the tangent is discontinuous; nearby teeth may jump. C1 yields continuous direction; C2 yields continuous magnitude as well.

Example

```
>>> comb = my_wire.curvature_comb(count=200, max_tooth_size=2.0)
>>> show(my_wire, Curve(comb))
```

derivative_at(*position: float | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float], order: int = 2, position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>*) → Vector

Derivative At

Generate a derivative along the underlying curve.

Parameters

- **position** (*float | VectorLike*) – distance, parameter value or point
- **order** (*int*) – derivative order. Defaults to 2
- **position_mode** (*PositionMode, optional*) – position calculation mode. Defaults to PositionMode.PARAMETER.

Raises
ValueError – position must be a float or a point

Returns
position on the underlying curve

Return type
Vector

end_point() → Vector

The end point of this edge.

Note that circles may have identical start and end points.

classmethod extrude(*obj:* Shape, *direction: VectorLike*) → *Edge* | *Face* | *Shell* | *Solid* | *Compound*

Unused - only here because Mixin1D is a subclass of Shape

property is_closed: bool

Are the start and end points equal?

property is_forward: bool

Does the Edge/Wire loop forward or reverse

property is_interior: bool

Check if the edge is an interior edge.

An interior edge lies between surfaces that are part of the body (internal to the geometry) and does not form part of the exterior boundary.

Returns
True if the edge is an interior edge, False otherwise.

Return type
bool

property length: float

Edge or Wire length

location_at(*distance: float*, *position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>*, *frame_method: ~build123d.build_enums.FrameMethod = <FrameMethod.FRENET>*, *x_dir: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None*) → Location

Locations along curve

Generate a location along the underlying curve.

Parameters

- **distance** (*float*) – distance or parameter value
- **position_mode** (*PositionMode, optional*) – position calculation mode. Defaults to PositionMode.PARAMETER.
- **frame_method** (*FrameMethod, optional*) – moving frame calculation method. The FRENET frame can "twist" or flip unexpectedly, especially near flat spots. The CORRECTED frame behaves more like a "camera dolly" or sweep profile would — it's smoother and more stable. Defaults to FrameMethod.FRENET.
- **x_dir** (*VectorLike, optional*) – override the x_dir to help with plane creation along a 1D shape. Must be perpendicular to shapes tangent. Defaults to None.

Returns

A Location object representing local coordinate system
at the specified distance.

Return type
Location

locations(*distances: ~collections.abc.Iterable[float], position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>, frame_method: ~build123d.build_enums.FrameMethod = <FrameMethod.FRENET>, x_dir: ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] | None = None*) → list[Location]

Locations along curve

Generate location along the curve

Parameters

- **distances** (*Iterable[float]*) – distance or parameter values

- **position_mode** (*PositionMode, optional*) – position calculation mode. Defaults to PositionMode.PARAMETER.
- **frame_method** (*FrameMethod, optional*) – moving frame calculation method. Defaults to FrameMethod.FRENET.
- **x_dir** (*VectorLike, optional*) – override the x_dir to help with plane creation along a 1D shape. Must be perpendicular to shapes tangent. Defaults to None.

Returns

A list of Location objects representing local coordinate
systems at the specified distances.

Return type
list[Location]

normal() → Vector

Calculate the normal Vector. Only possible for planar curves.

Returns
normal vector

Args:

Returns:

offset_2d(*distance: float*, *kind: ~build123d.build_enums.Kind = <Kind.ARC>*, *side: ~build123d.build_enums.Side = <Side.BOTH>*, *closed: bool = True*) → *Edge* | *Wire*

2d Offset

Offsets a planar edge/wire

Parameters

- **distance** (*float*) – distance from edge/wire to offset
- **kind** (Kind, *optional*) – offset corner transition. Defaults to Kind.ARC.
- **side** (*Side, optional*) – side to place offset. Defaults to Side.BOTH.
- **closed** (*bool, optional*) – if Side!=BOTH, close the LEFT or RIGHT offset. Defaults to True.

Raises

- **RuntimeError** – Multiple Wires generated
- **RuntimeError** – Unexpected result type

Returns
offset wire

Return type
Wire

perpendicular_line(*length: float*, *u_value: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Edge*

Create a line on the given plane perpendicular to and centered on beginning of self

Parameters

- **length** (*float*) – line length
- **u_value** (*float*) – position along line between 0.0 and 1.0

- **plane** (Plane, *optional*) – plane containing perpendicular line. Defaults to Plane.XY.

Returns
perpendicular line

Return type
Edge

position_at(*position: float*, *position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>*) → Vector

Position At

Generate a position along the underlying Wire.

Parameters

- **position** (*float*) – distance or parameter value
- **position_mode** (*PositionMode, optional*) – position calculation mode. Defaults to PositionMode.PARAMETER.

Returns
position on the underlying curve

Return type
Vector

positions(*distances: ~collections.abc.Iterable[float] | None = None*, *position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>*, *deflection: float | None = None*) → list[Vector]

Positions along curve

Generate positions along the underlying curve

Parameters

- **distances** (*Iterable[float] | None, optional*) – distance or parameter values. Defaults to None.
- **position_mode** (*PositionMode, optional*) – position calculation mode only applies when using distances. Defaults to PositionMode.PARAMETER.
- **deflection** (*float | None, optional*) – maximum deflection between the curve and the polygon that results from the computed points. Defaults to None.

Returns
positions along curve

Return type
list[Vector]

project(*face:* Face, *direction: VectorLike*, *closest: bool = True*) → *Edge* | *Wire* | *ShapeList*[*Edge* | *Wire*]

Project onto a face along the specified direction

Parameters

- **face** – Face:
- **direction** – VectorLike:
- **closest** – bool: (Default value = True)

Returns:

project_to_viewport(*viewport_origin: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *viewport_up: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 1)*, *look_at: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *focus: float | None = None*) → tuple[*ShapeList*[*Edge*], *ShapeList*[*Edge*]]

Project a shape onto a viewport returning visible and hidden Edges.

Parameters

- **viewport_origin** (*VectorLike*) – location of viewport
- **viewport_up** (*VectorLike, optional*) – direction of the viewport y axis. Defaults to (0, 0, 1).
- **look_at** (*VectorLike, optional*) – point to look at. Defaults to None (center of shape).
- **focus** (*float, optional*) – the focal length for perspective projection Defaults to None (orthographic projection)

Returns
visible & hidden Edges

Return type
tuple[ShapeList[Edge],ShapeList[Edge]]

property radius: float

Calculate the radius.

Note that when applied to a Wire, the radius is simply the radius of the first edge.

Args:

Returns
radius

Raises
ValueError – if kernel can not reduce the shape to a circular edge

start_point() → Vector

The start point of this edge

Note that circles may have identical start and end points.

tangent_angle_at(*location_param: float = 0.5*, *position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>*, *plane: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → float

Compute the tangent angle at the specified location

Parameters

- **location_param** (*float, optional*) – distance or parameter value. Defaults to 0.5.
- **position_mode** (*PositionMode, optional*) – position calculation mode. Defaults to PositionMode.PARAMETER.
- **plane** (*Plane, optional*) – plane line was constructed on. Defaults to Plane.XY.

Returns
angle in degrees between 0 and 360

Return type
float

tangent_at(*position: float | ~build123d.geometry.Vector | tuple[float, float] | tuple[float, float, float] | ~collections.abc.Sequence[float] = 0.5*, *position_mode: ~build123d.build_enums.PositionMode = <PositionMode.PARAMETER>*) → Vector

Find the tangent at a given position on the 1D shape where the position is either a float (or int) parameter or a point that lies on the shape.

Parameters

- **position** (*float | VectorLike*) – distance, parameter value, or point on shape. Defaults to 0.5.
- **position_mode** (*PositionMode, optional*) – position calculation mode. Defaults to PositionMode.PARAMETER.

Returns
tangent value

Return type
Vector

property volume: float

volume - the volume of this Edge or Wire, which is always zero

class Mixin2D(*obj: TopoDS_Shape | None = None*, *label: str = ''*, *color: ColorLike | None = None*, *parent: Compound | None = None*)

Additional methods to add to Face and Shell class

classmethod cast(*obj: TopoDS_Shape*) → *Vertex* | *Edge* | *Wire* | *Face* | *Shell*

Returns the right type of wrapper, given a OCCT object

classmethod extrude(*obj:* Shape, *direction: VectorLike*) → *Edge* | *Face* | *Shell* | *Solid* | *Compound*

Unused - only here because Mixin1D is a subclass of Shape

find_intersection_points(*other: Axis*, *tolerance: float = 1e-06*) → list[tuple[Vector, Vector]]

Find point and normal at intersection

Return both the point(s) and normal(s) of the intersection of the axis and the shape

Parameters
axis (Axis) – axis defining the intersection line

Returns
Point and normal of intersection

Return type
list[*tuple*[Vector, Vector]]

abstract location_at(**args: Any*, ***kwargs: Any*) → Location

A location from a face or shell

offset(*amount: float*) → Self

Return a copy of self moved along the normal by amount

project_to_viewport(*viewport_origin: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *viewport_up: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 1)*, *look_at: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *focus: float | None = None*) → tuple[*ShapeList*[*Edge*], *ShapeList*[*Edge*]]

Project a shape onto a viewport returning visible and hidden Edges.

Parameters

- **viewport_origin** (*VectorLike*) – location of viewport
- **viewport_up** (*VectorLike, optional*) – direction of the viewport y axis. Defaults to (0, 0, 1).
- **look_at** (*VectorLike, optional*) – point to look at. Defaults to None (center of shape).
- **focus** (*float, optional*) – the focal length for perspective projection Defaults to None (orthographic projection)

Returns
visible & hidden Edges

Return type
tuple[ShapeList[Edge],ShapeList[Edge]]

split_by_perimeter(*perimeter:* Edge | Wire, *keep: Literal[Keep.INSIDE, Keep.OUTSIDE]*) → *Face* | *Shell* | *ShapeList*[*Face*] | None

split_by_perimeter(*perimeter:* Edge | Wire, *keep: Literal[Keep.BOTH]*) → tuple[*Face* | *Shell* | *ShapeList*[*Face*] | None, *Face* | *Shell* | *ShapeList*[*Face*] | None]

split_by_perimeter(*perimeter:* Edge | Wire, *keep: Literal[Keep.INSIDE] = Keep.INSIDE*) → *Face* | *Shell* | *ShapeList*[*Face*] | None

split_by_perimeter

Divide the faces of this object into those within the perimeter and those outside the perimeter.

Note: this method may fail if the perimeter intersects shape edges.

Parameters

- **perimeter** (*Union[Edge,Wire]*) – closed perimeter
- **keep** (Keep, *optional*) – which object(s) to return. Defaults to Keep.INSIDE.

Raises

- **ValueError** – perimeter must be closed
- **ValueError** – keep must be one of Keep.INSIDE|OUTSIDE|BOTH

Returns

Union[Face | Shell | ShapeList[Face] | None, Tuple[Face | Shell | ShapeList[Face] | None]: The result of the split operation.

- **Keep.INSIDE**: Returns the inside part as a *Shell* or *Face*, or *None* if no inside part is found.
- **Keep.OUTSIDE**: Returns the outside part as a *Shell* or *Face*, or *None* if no outside part is found.
- **Keep.BOTH**: Returns a tuple *(inside, outside)* where each element is either a *Shell*, *Face*, or *None* if no corresponding part is found.

touch(*other:* Shape, *tolerance: float = 1e-06*, *found_faces:* ShapeList | *None = None*, *found_edges:* ShapeList | *None = None*) → *ShapeList*

Find boundary contacts between this 2D shape and another shape.

Returns the highest-dimensional contact at each location, filtered to avoid returning lower-dimensional boundaries of higher-dimensional contacts.

For Face/Shell: - Face + Face → Vertex (shared corner or crossing point without edge/face overlap) - Face + Edge/Vertex → no touch (intersect already returns dim 0)

Parameters

- **other** – Shape to find contacts with
- **tolerance** – tolerance for contact detection
- **found_faces** – pre-found faces to filter against (from Mixin3D.touch)
- **found_edges** – pre-found edges to filter against (from Mixin3D.touch)

Returns
ShapeList of contact shapes (Vertex only for 2D+2D)

class **Mixin3D**(*obj: TopoDS_Shape | None = None*, *label: str = ''*, *color: ColorLike | None = None*, *parent: Compound | None = None*)

Additional methods to add to 3D Shape classes

classmethod **cast**(*obj: TopoDS_Shape*) → Self

Returns the right type of wrapper, given a OCCT object

center(*center_of: ~build123d.build_enums.CenterOf = <CenterOf.MASS>*) → Vector

Return center of object

Find center of object

Parameters
center_of (*CenterOf, optional*) – center option. Defaults to CenterOf.MASS.

Raises

- **ValueError** – Center of GEOMETRY is not supported for this object
- **NotImplementedError** – Unable to calculate center of mass of this object

Returns
center

Return type
Vector

chamfer(*length: float*, *length2: float | None*, *edge_list: Iterable[Edge]*, *face: Face | None = None*) → *Solid | Part*

Chamfer

Chamfers the specified edges of this solid.

Parameters

- **length** (*float*) – length > 0, the length (length) of the chamfer
- **length2** (*Optional[float]*) – length2 > 0, optional parameter for asymmetrical chamfer. Should be *None* if not required.
- **edge_list** (*Iterable[Edge]*) – a list of Edge objects, which must belong to this solid
- **face** (*Face, optional*) – identifies the side where length is measured. The edge(s) must be part of the face

Returns
Chamfered solid or 3D composite

Return type
Solid | Part

dprism(*basis: Face | None*, *bounds: list[Face | Wire]*, *depth: float | None = None*, *taper: float = 0*, *up_to_face: Face | None = None*, *thru_all: bool = True*, *additive: bool = True*) → *Solid*

Make a prismatic feature (additive or subtractive)

Parameters

- **basis** (*Optional[Face]*) – face to perform the operation on
- **bounds** (*list[Union[Face,Wire]]*) – list of profiles
- **depth** (*float, optional*) – depth of the cut or extrusion. Defaults to None.
- **taper** (*float, optional*) – in degrees. Defaults to 0.
- **up_to_face** (*Face, optional*) – a face to extrude until. Defaults to None.
- **thru_all** (*bool, optional*) – cut thru_all. Defaults to True.
- **additive** (*bool, optional*) – Defaults to True.

Returns
prismatic feature

Return type
Solid

classmethod extrude(*obj: Shape*, *direction: VectorLike*) → *Edge | Face | Shell | Solid | Compound*

Unused - only here because Mixin1D is a subclass of Shape

fillet(*radius: float*, *edge_list: Iterable[Edge]*) → *Solid | Part*

Fillet

Fillets the specified edges of this solid.

Parameters

- **radius** (*float*) – float > 0, the radius of the fillet
- **edge_list** (*Iterable[Edge]*) – a list of Edge objects, which must belong to this solid

Returns
Filleted solid or 3D composite

Return type
Solid | Part

find_intersection_points(*other: Axis*, *tolerance: float = 1e-06*) → list[tuple[Vector, Vector]]

Find point and normal at intersection

Return both the point(s) and normal(s) of the intersection of the axis and the shape

Parameters
axis (*Axis*) – axis defining the intersection line

Returns
Point and normal of intersection

Return type
list[*tuple*[Vector, Vector]]

hollow(*faces: ~collections.abc.Iterable[~topology.two_d.Face] | None*, *thickness: float*, *tolerance: float = 0.0001*, *kind: ~build123d.build_enums.Kind = <Kind.ARC>*) → *Solid*

Hollow

Return the outer shelled solid of self.

Parameters

- **faces** (*Optional[Iterable[Face]]*) – faces to be removed,
- **list.** (*which must be part of the solid. Can be an empty*)
- **thickness** (*float*) – shell thickness - positive shells outwards, negative shells inwards.
- **tolerance** (*float, optional*) – modelling tolerance of the method. Defaults to 0.0001.
- **kind** (*Kind, optional*) – intersection type. Defaults to Kind.ARC.

Raises

ValueError – Kind.TANGENT not supported

Returns

A hollow solid.

Return type

Solid

is_inside(*point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *tolerance: float = 1e-06*) → bool

Returns whether or not the point is inside a solid or compound object within the specified tolerance.

Parameters

- **point** – tuple or Vector representing 3D point to be tested
- **tolerance** – tolerance for inside determination, default=1.0e-6
- **point** – VectorLike:
- **tolerance** – float: (Default value = 1.0e-6)

Returns

bool indicating whether or not point is within solid

max_fillet(*edge_list: Iterable[Edge]*, *tolerance=0.1*, *max_iterations: int = 10*) → float

Find Maximum Fillet Size

Find the largest fillet radius for the given Shape and edges with a recursive binary search.

Example

max_fillet_radius = my_shape.max_fillet(shape_edges) max_fillet_radius = my_shape.max_fillet(shape_edges, tolerance=0.5, max_iterations=8)

Parameters

- **edge_list** (*Iterable[Edge]*) – a sequence of Edge objects, which must belong to this solid
- **tolerance** (*float, optional*) – maximum error from actual value. Defaults to 0.1.
- **max_iterations** (*int, optional*) – maximum number of recursive iterations. Defaults to 10.

Raises

- **RuntimeError** – failed to find the max value
- **ValueError** – the provided Shape is invalid

Returns

maximum fillet radius

Return type
float

offset_3d(*openings: ~collections.abc.Iterable[~topology.two_d.Face] | None*, *thickness: float*, *tolerance: float = 0.0001*, *kind: ~build123d.build_enums.Kind = <Kind.ARC>*) → *Solid*

Shell

Make an offset solid of self.

Parameters

- **openings** (*Optional[Iterable[Face]]*) – faces to be removed, which must be part of the solid. Can be an empty list.
- **thickness** (*float*) – offset amount - positive offset outwards, negative inwards
- **tolerance** (*float, optional*) – modelling tolerance of the method. Defaults to 0.0001.
- **kind** (*Kind, optional*) – intersection type. Defaults to Kind.ARC.

Raises
ValueError – Kind.TANGENT not supported

Returns
A shelled solid.

Return type
Solid

project_to_viewport(*viewport_origin: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *viewport_up: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 1)*, *look_at: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*, *focus: float | None = None*) → tuple[*ShapeList*[*Edge*], *ShapeList*[*Edge*]]

Project a shape onto a viewport returning visible and hidden Edges.

Parameters

- **viewport_origin** (*VectorLike*) – location of viewport
- **viewport_up** (*VectorLike, optional*) – direction of the viewport y axis. Defaults to (0, 0, 1).
- **look_at** (*VectorLike, optional*) – point to look at. Defaults to None (center of shape).
- **focus** (*float, optional*) – the focal length for perspective projection Defaults to None (orthographic projection)

Returns
visible & hidden Edges

Return type
tuple[ShapeList[Edge],ShapeList[Edge]]

class Shape(*obj: TopoDS_Shape | None = None*, *label: str = ''*, *color: ColorLike | None = None*, *parent: Compound | None = None*)

Base class for all CAD objects such as Edge, Face, Solid, etc.

Parameters

- **obj** (*TopoDS_Shape, optional*) – OCCT object. Defaults to None.
- **label** (*str, optional*) – Defaults to ‘’.

- **color** (*ColorLike, optional*) – Defaults to None.
- **parent** (Compound, *optional*) – assembly parent. Defaults to None.

Variables

- **wrapped** (*TopoDS_Shape*) – the OCP object
- **label** (*str*) – user assigned label
- **color** (Color) – object color
- **(dict[str** (*joints*) – Joint]): dictionary of joints bound to this object (Solid only)
- **children** (Shape) – list of assembly children of this object (Compound only)
- **topo_parent** (Shape) – assembly parent of this object

__add__(*other: None*) → Self

__add__(*other:* Shape | *Iterable[*Shape*]*) → Self | *Compound*

fuse shape to self operator +

__and__(*other:* Shape | *Iterable[*Shape*]*) → None | Self | *Compound*

intersect shape with self operator &

__copy__() → Self

Return shallow copy or reference of self

Create an copy of this Shape that shares the underlying TopoDS_TShape.

Used when there is a need for many objects with the same CAD structure but at different Locations, etc. - for examples fasteners in a larger assembly. By sharing the TopoDS_TShape, the memory size of such assemblies can be greatly reduced.

Changes to the CAD structure of the base object will be reflected in all instances.

__deepcopy__(*memo*) → Self

Return deepcopy of self

__eq__(*other*) → bool

Check if two shapes are the same.

This method checks if the current shape is the same as the other shape. Two shapes are considered the same if they share the same TShape with the same Locations. Orientations may differ.

Parameters
other (Shape) – The shape to compare with.

Returns
True if the shapes are the same, False otherwise.

Return type
bool

__hash__() → int

Return hash code

__rmul__(*other: Plane | Location*) → Self

__rmul__(*other: Iterable[Plane | Location]*) → list[Self]

right multiply for positioning operator *

__sub__(*other: None*) → Self

__sub__(*other:* Shape | *Iterable[*Shape*]*) → Self | *Compound*

cut shape from self operator -

property area: float

area -the surface area of all faces in this Shape

bounding_box(*tolerance: float | None = None*, *optimal: bool = True*) → BoundBox

Create a bounding box for this Shape.

Parameters
tolerance (*float, optional*) – Defaults to None.

Returns
A box sized to contain this Shape

Return type
BoundBox

abstract classmethod cast(*obj: TopoDS_Shape*) → Self

Returns the right type of wrapper, given a OCCT object

clean() → Self

Remove internal edges

Returns
Original object with extraneous internal edges removed

Return type
Shape

closest_points(*other:* Shape | *Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → tuple[Vector, Vector]

Points on two shapes where the distance between them is minimal

property color: None | Color

Get the shape's color. If it's None, get the color of the nearest ancestor, assign it to this Shape and return this value.

static combined_center(*objects: ~collections.abc.Iterable[~topology.shape_core.Shape], center_of: ~build123d.build_enums.CenterOf = <CenterOf.MASS>*) → Vector

combined center

Calculates the center of a multiple objects.

Parameters

- **objects** (*Iterable[*Shape*]*) – list of objects
- **center_of** (CenterOf, *optional*) – centering option. Defaults to CenterOf.MASS.

Raises
ValueError – CenterOf.GEOMETRY not implemented

Returns
center of multiple objects

Return type
Vector

composite_factories: ClassVar[dict[int | None, Callable[[Iterable[*Shape*]], *Shape*]]] = {1: <class 'topology.composite.Curve'>, 2: <class 'topology.composite.Sketch'>, 3: <class 'topology.composite.Part'>, None: <class 'topology.composite.Compound'>}

compound() → *Compound*

Return the Compound

compounds() → *ShapeList*[*Compound*]

compounds - all the compounds in this Shape

static compute_mass(*obj:* Shape) → float

Calculates the 'mass' of an object.

Parameters

- **obj** – Compute the mass of this object
- **obj** – Shape:

Returns:

copy_attributes_to(*target:* Shape, *exceptions: Iterable[str] | None = None*)

Copy common object attributes to target

Note that preset attributes of target will not be overridden.

Parameters

- **target** (Shape) – object to gain attributes
- **exceptions** (*Iterable[str], optional*) – attributes not to copy

Raises

ValueError – invalid attribute

cut(**to_cut:* Shape) → Self | *Compound*

Remove the positional arguments from this Shape.

Parameters

***to_cut** – Shape:

Returns

Resulting object may be of a different class than self

Return type

Self | Compound

distance(*other:* Shape) → float

Minimal distance between two shapes

Parameters

other – Shape:

Returns:

distance_to(*other:* Shape | *Vector* | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*) → float

Minimal distance between two shapes

distance_to_with_closest_points(*other:* Shape | *Vector* | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*) → tuple[float, Vector, Vector]

Minimal distance between two shapes and the points on each shape

distances(**others:* Shape) → Iterator[float]

Minimal distances to between self and other shapes

Parameters

***others** – Shape:

Returns:

downcast_LUT = {<TopAbs_ShapeEnum.TopAbs_COMPOUND: 0>: <built-in method Compound of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_COMPSOLID: 1>: <built-in method CompSolid of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_EDGE: 6>: <built-in method Edge of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_FACE: 4>: <built-in method Face of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_SHELL: 3>: <built-in method Shell of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_SOLID: 2>: <built-in method Solid of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_VERTEX: 7>: <built-in method Vertex of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_WIRE: 5>: <built-in method Wire of PyCapsule object>}

edge() → *Edge*

Return the Edge

edges() → *ShapeList*[*Edge*]

edges - all the edges in this Shape - subclasses may override

entities(*topo_type: Literal['Vertex', 'Edge', 'Wire', 'Face', 'Shell', 'Solid', 'Compound']*) → list[TopoDS_Shape]

Return all of the TopoDS sub entities of the given type

abstract classmethod extrude(*obj:* Shape, *direction: VectorLike*) → *Edge* | *Face* | *Shell* | *Solid* | *Compound*

Extrude a Shape in the provided direction. * Vertices generate Edges * Edges generate Faces * Wires generate Shells * Faces generate Solids * Shells generate Compounds

Parameters
direction (*VectorLike*) – direction and magnitude of extrusion

Raises

- **ValueError** – Unsupported class
- **RuntimeError** – Generated invalid result

Returns
extruded shape

Return type
Edge | Face | Shell | Solid | Compound

face() → *Face*

Return the Face

faces() → *ShapeList*[*Face*]

faces - all the faces in this Shape

faces_intersected_by_axis(*axis:* Axis, *tol: float = 0.0001*) → *ShapeList*[*Face*]

Line Intersection

Computes the intersections between the provided axis and the faces of this Shape

Parameters

- **axis** (*Axis*) – Axis on which the intersection line rests
- **tol** (*float, optional*) – Intersection tolerance. Defaults to 1e-4.

Returns
A list of intersected faces sorted by distance from axis.position

Return type
list[Face]

fix() → Self

fix - try to fix shape if not valid

fuse(**to_fuse:* Shape, *glue: bool = False*, *tol: float | None = None*) → Self | *Compound*

Fuse a sequence of shapes into a single shape.

Parameters

- **to_fuse** (*sequence Shape*) – shapes to fuse
- **glue** (*bool, optional*) – performance improvement for some shapes. Defaults to False.
- **tol** (*float, optional*) – tolerance. Defaults to None.

Returns
Resulting object may be of a different class than self

Return type
Self | Compound

geom_LUT_EDGE: dict[GeomAbs_CurveType, GeomType] =
{<GeomAbs_CurveType.GeomAbs_BSplineCurve: 6>: <GeomType.BSPLINE>,
<GeomAbs_CurveType.GeomAbs_BezierCurve: 5>: <GeomType.BEZIER>,
<GeomAbs_CurveType.GeomAbs_Circle: 1>: <GeomType.CIRCLE>,
<GeomAbs_CurveType.GeomAbs_Ellipse: 2>: <GeomType.ELLIPSE>,
<GeomAbs_CurveType.GeomAbs_Hyperbola: 3>: <GeomType.HYPERBOLA>,
<GeomAbs_CurveType.GeomAbs_Line: 0>: <GeomType.LINE>,
<GeomAbs_CurveType.GeomAbs_OffsetCurve: 7>: <GeomType.OFFSET>,
<GeomAbs_CurveType.GeomAbs_OtherCurve: 8>: <GeomType.OTHER>,
<GeomAbs_CurveType.GeomAbs_Parabola: 4>: <GeomType.PARABOLA>}

geom_LUT_FACE: dict[GeomAbs_SurfaceType, GeomType] =
{<GeomAbs_SurfaceType.GeomAbs_BSplineSurface: 6>: <GeomType.BSPLINE>,
<GeomAbs_SurfaceType.GeomAbs_BezierSurface: 5>: <GeomType.BEZIER>,
<GeomAbs_SurfaceType.GeomAbs_Cone: 2>: <GeomType.CONE>,
<GeomAbs_SurfaceType.GeomAbs_Cylinder: 1>: <GeomType.CYLINDER>,
<GeomAbs_SurfaceType.GeomAbs_OffsetSurface: 9>: <GeomType.OFFSET>,
<GeomAbs_SurfaceType.GeomAbs_OtherSurface: 10>: <GeomType.OTHER>,
<GeomAbs_SurfaceType.GeomAbs_Plane: 0>: <GeomType.PLANE>,
<GeomAbs_SurfaceType.GeomAbs_Sphere: 3>: <GeomType.SPHERE>,
<GeomAbs_SurfaceType.GeomAbs_SurfaceOfExtrusion: 8>: <GeomType.EXTRUSION>,
<GeomAbs_SurfaceType.GeomAbs_SurfaceOfRevolution: 7>: <GeomType.REVOLUTION>,
<GeomAbs_SurfaceType.GeomAbs_Torus: 4>: <GeomType.TORUS>}

property geom_type: GeomType

Gets the underlying geometry type.

Returns
The geometry type of the shape

Return type
GeomType

static get_shape_list(*shape:* Shape, *entity_type: Literal['Vertex']*) → *ShapeList*[*Vertex*]

static get_shape_list(*shape:* Shape, *entity_type: Literal['Edge']*) → *ShapeList*[*Edge*]

static get_shape_list(*shape:* Shape, *entity_type: Literal['Wire']*) → *ShapeList*[*Wire*]

static get_shape_list(*shape:* Shape, *entity_type: Literal['Face']*) → *ShapeList*[*Face*]
static get_shape_list(*shape:* Shape, *entity_type: Literal['Shell']*) → *ShapeList*[*Shell*]
static get_shape_list(*shape:* Shape, *entity_type: Literal['Solid']*) → *ShapeList*[*Solid*]
static get_shape_list(*shape:* Shape, *entity_type: Literal['Compound']*) → *ShapeList*[*Compound*]

Helper to extract entities of a specific type from a shape.

static get_single_shape(*shape:* Shape, *entity_type: Literal['Vertex']*) → *Vertex*
static get_single_shape(*shape:* Shape, *entity_type: Literal['Edge']*) → *Edge*
static get_single_shape(*shape:* Shape, *entity_type: Literal['Wire']*) → *Wire*
static get_single_shape(*shape:* Shape, *entity_type: Literal['Face']*) → *Face*
static get_single_shape(*shape:* Shape, *entity_type: Literal['Shell']*) → *Shell*
static get_single_shape(*shape:* Shape, *entity_type: Literal['Solid']*) → *Solid*
static get_single_shape(*shape:* Shape, *entity_type: Literal['Compound']*) → *Compound*

Return the single entity of the requested type.

Raises
ValueError – if the number of matching entities is not exactly one.

get_top_level_shapes() → *ShapeList*[*Shape*]

Retrieve the first level of child shapes from the shape.

This method collects all the non-compound shapes directly contained in the current shape. If the wrapped shape is a *TopoDS_Compound*, it traverses its immediate children and collects all shapes that are not further nested compounds. Nested compounds are traversed to gather their non-compound elements without returning the nested compound itself.

Returns
A list of all first-level non-compound child shapes.

Return type
ShapeList[Shape]

Example

If the current shape is a compound containing both simple shapes (e.g., edges, vertices) and other compounds, the method returns a list of only the simple shapes directly contained at the top level.

property global_location: Location

The location of this Shape relative to the global coordinate system.

This property computes the composite transformation by traversing the hierarchy from the root of the assembly to this node, combining the location of each ancestor. It reflects the absolute position and orientation of the shape in world space, even when the shape is deeply nested within an assembly.

Note

This is only meaningful when the Shape is part of an assembly tree where parent-child relationships define relative placements.

intersect(**to_intersect:* Shape | *Vector* | *Location* | *Axis* | *Plane*, *tolerance: float = 1e-06*, *include_touched: bool = False*) → *ShapeList* | None

Find where bodies/interiors meet (overlap or crossing geometry).

This is the main entry point for intersection operations. Handles geometry conversion and delegates to subclass _intersect() implementations.

Semantics:

- Multiple arguments use AND (chaining): c.intersect(s1, s2) = c ∩ s1 ∩ s2
- Compound arguments use OR (distribution): c.intersect(Compound([s1, s2])) = (c ∩ s1) ∪ (c ∩ s2)

Parameters

- **to_intersect** – Shape(s) or geometry objects to intersect with
- **tolerance** – tolerance for intersection detection
- **include_touched** – if True, include boundary contacts without interior overlap (only relevant when Solids are involved)

Returns

ShapeList of intersection results, or None if no intersection

inverse_shape_LUT = {'CompSolid': <TopAbs_ShapeEnum.TopAbs_COMPSOLID: 1>, 'Compound': <TopAbs_ShapeEnum.TopAbs_COMPOUND: 0>, 'Edge': <TopAbs_ShapeEnum.TopAbs_EDGE: 6>, 'Face': <TopAbs_ShapeEnum.TopAbs_FACE: 4>, 'Shell': <TopAbs_ShapeEnum.TopAbs_SHELL: 3>, 'Solid': <TopAbs_ShapeEnum.TopAbs_SOLID: 2>, 'Vertex': <TopAbs_ShapeEnum.TopAbs_VERTEX: 7>, 'Wire': <TopAbs_ShapeEnum.TopAbs_WIRE: 5>}

is_equal(*other:* Shape) → bool

Returns True if two shapes are equal, i.e. if they share the same TShape with the same Locations and Orientations. Also see *is_same()*.

Parameters

other – Shape:

Returns:

property is_manifold: bool

Check if each edge in the given Shape has exactly two faces associated with it (skipping degenerate edges). If so, the shape is manifold.

Returns

is the shape manifold or water tight

Return type

bool

property is_null: bool

Returns true if this shape is null. In other words, it references no underlying shape with the potential to be given a location and an orientation.

property is_planar_face: bool

Is the shape a planar face even though its geom_type may not be PLANE

is_same(*other:* Shape) → bool

Returns True if other and this shape are same, i.e. if they share the same TShape with the same Locations. Orientations may differ. Also see *is_equal()*

Parameters

other – Shape:

Returns:

property is_valid: bool

Returns True if no defect is detected on the shape S or any of its subshapes. See the OCCT docs on BRepCheck_Analyzer::IsValid for a full description of what is checked.

locate(*loc: Location*) → Self

Apply a location in absolute sense to self

Parameters
loc – Location:

Returns:

located(*loc: Location*) → Self

Apply a location in absolute sense to a copy of self

Parameters
loc (Location) – new absolute location

Returns
copy of Shape at location

Return type
Shape

property location: Location

Get this Shape's Location

classmethod make_composite(*shapes: Iterable[*Shape*]*, *dimension: int | None = None*) → *Shape*

Build the registered composite for a dimension.

property matrix_of_inertia: list[list[float]]

Compute the inertia matrix (moment of inertia tensor) of the shape.

The inertia matrix represents how the mass of the shape is distributed with respect to its reference frame. It is a 3×3 symmetric tensor that describes the resistance of the shape to rotational motion around different axes.

Returns

A 3×3 nested list representing the inertia matrix. The elements of the matrix are given as:

Ixx Ixy Ixz |
Ixy Iyy Iyz |
Ixz Iyz Izz |

where: - Ixx, Iyy, Izz are the moments of inertia about the X, Y, and Z axes. - Ixy, Ixz, Iyz are the products of inertia.

Return type
list[*list*[*float*]]

Example

```
>>> obj = MyShape()
>>> obj.matrix_of_inertia
[[1000.0, 50.0, 0.0],
```

(continues on next page)

(continued from previous page)

```
[50.0, 1200.0, 0.0],
[0.0, 0.0, 300.0]]
```

Notes

- The inertia matrix is computed relative to the shape's center of mass.
- It is commonly used in structural analysis, mechanical simulations, and physics-based motion calculations.

mesh(*tolerance: float*, *angular_tolerance: float = 0.1*)

Generate triangulation if none exists.

Parameters

- **tolerance** – float:
- **angular_tolerance** – float: (Default value = 0.1)

Returns:

mirror(*mirror_plane: Plane | None = None*) → Self

Applies a mirror transform to this Shape. Does not duplicate objects about the plane.

Parameters
mirror_plane (Plane) – The plane to mirror about. Defaults to Plane.XY

Returns
The mirrored shape

move(*loc: Location*) → Self

Apply a location in relative sense (i.e. update current location) to self

Parameters
loc – Location:

Returns:

moved(*loc: Location | Plane*) → Self

Apply a location in relative sense (i.e. update current location) to a copy of self

Parameters
loc (Location | Plane) – new location relative to current location

Returns
copy of Shape moved to relative location

Return type
Shape

property orientation: Vector

Get the orientation component of this Shape's Location

oriented_bounding_box() → OrientedBoundBox

Create an oriented bounding box for this Shape.

Returns
A box oriented and sized to contain this Shape

Return type
OrientedBoundBox

property position: Vector

Get the position component of this Shape's Location

property principal_properties: list[tuple[Vector, float]]

Compute the principal moments of inertia and their corresponding axes.

Returns

A list of tuples, where each tuple contains: - A *Vector* representing the axis of inertia. - A *float* representing the moment of inertia for that axis.

Return type

list[*tuple*[Vector, *float*]]

Example

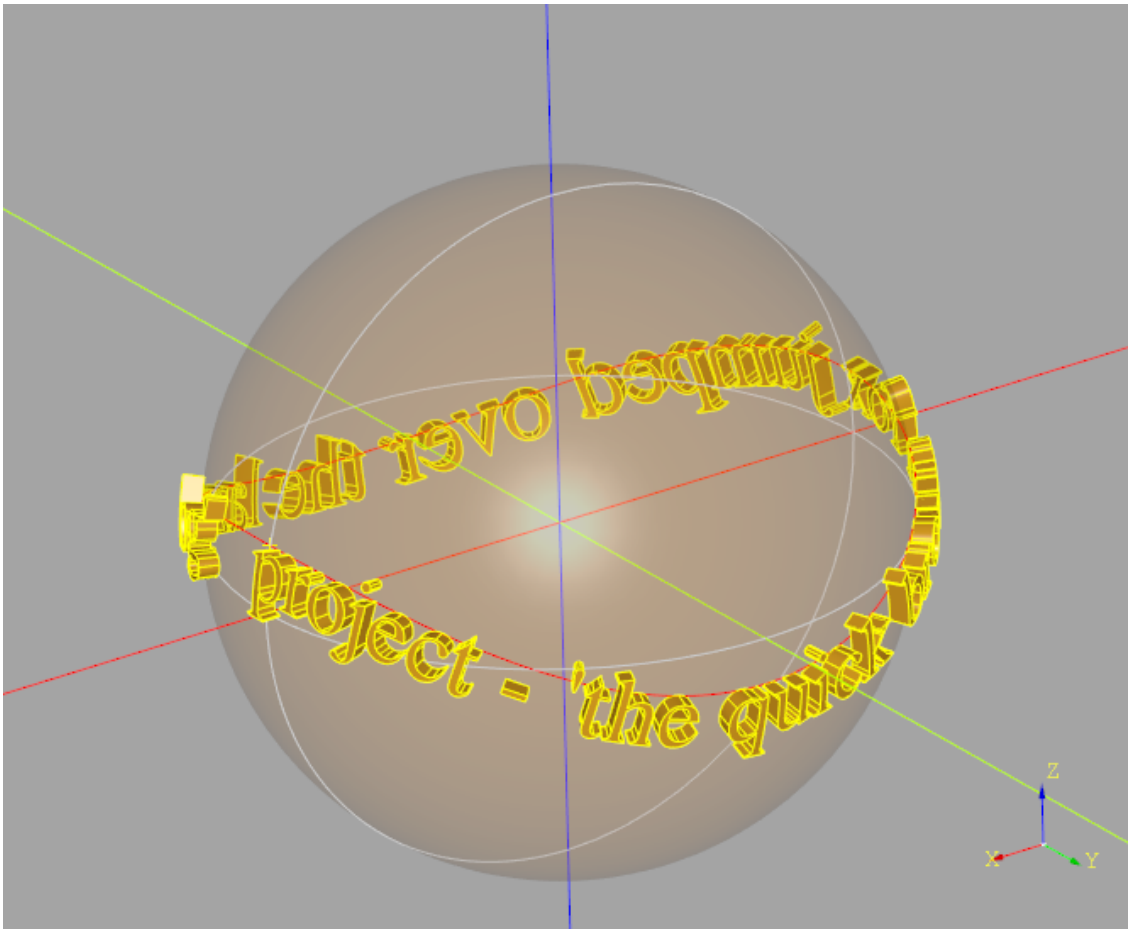
```
>>> obj = MyShape()
>>> obj.principal_properties
[(Vector(1, 0, 0), 1200.0),
(Vector(0, 1, 0), 1000.0),
(Vector(0, 0, 1), 300.0)]
```

project_faces(*faces: list[Face] | Compound*, *path: Wire | Edge*, *start: float = 0*) → *ShapeList*[*Face*]

Projected Faces following the given path on Shape

Project by positioning each face of to the shape along the path and projecting onto the surface.

Note that projection may result in distortion depending on the shape at a position along the path.



Parameters

- **faces** (*Union[list[Face], Compound]*) – faces to project
- **path** – Path on the Shape to follow
- **start** – Relative location on path to start the faces. Defaults to 0.

Returns
The projected faces

radius_of_gyration(*axis: Axis*) → float

Compute the radius of gyration of the shape about a given axis.

The radius of gyration represents the distance from the axis at which the entire mass of the shape could be concentrated without changing its moment of inertia. It provides insight into how mass is distributed relative to the axis and is useful in structural analysis, rotational dynamics, and mechanical simulations.

Parameters
axis (Axis) – The axis about which the radius of gyration is computed. The axis should be defined in the same coordinate system as the shape.

Returns
The radius of gyration in the same units as the shape's dimensions.

Return type
float

Example

```
>>> obj = MyShape()
>>> axis = Axis((0, 0, 0), (0, 0, 1))
>>> obj.radius_of_gyration(axis)
5.47
```

Notes

- The radius of gyration is computed based on the shape's mass properties.
- It is useful for evaluating structural stability and rotational behavior.

classmethod register_composite_factory(*dimension: int | None*, *factory: Callable[[Iterable[Shape]], Shape]*) → None

Register a composite constructor without importing it here.

relocate(*loc: Location*)

Change the location of self while keeping it geometrically similar

Parameters
loc (Location) – new location to set for self

rotate(*axis: Axis*, *angle: float*, *transform: bool = False*) → Self

rotate a copy

Rotates a shape around an axis.

Parameters

- **axis** (Axis) – rotation Axis
- **angle** (*float*) – angle to rotate, in degrees
- **transform** (*bool*) – regenerate the shape instead of just changing its location. Defaults to False.

Returns
a copy of the shape, rotated

scale(*factor: float | tuple[float, float, float]*, *about: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → Self

Scale this shape about a point.

Non-uniform scaling may change the underlying geometry type to splines. When `about` isn't provided, the shape is scaled about its location.

Parameters

- **factor** (*float | tuple[float, float, float]*) – uniform scale factor or three scale factors for the X, Y and Z directions.
- **about** (*VectorLike, optional*) – point to scale about. Defaults to the shape's location position.

Returns
a copy of the scaled shape.

Return type
Shape

shape_LUT = {<TopAbs_ShapeEnum.TopAbs_COMPOUND: 0>: 'Compound', <TopAbs_ShapeEnum.TopAbs_COMPSOLID: 1>: 'CompSolid', <TopAbs_ShapeEnum.TopAbs_EDGE: 6>: 'Edge', <TopAbs_ShapeEnum.TopAbs_FACE: 4>: 'Face', <TopAbs_ShapeEnum.TopAbs_SHELL: 3>: 'Shell', <TopAbs_ShapeEnum.TopAbs_SOLID: 2>: 'Solid', <TopAbs_ShapeEnum.TopAbs_VERTEX: 7>: 'Vertex', <TopAbs_ShapeEnum.TopAbs_WIRE: 5>: 'Wire'}

shape_properties_LUT: dict[TopAbs_ShapeEnum, Callable[[TopoDS_Shape, GProp_GProps], None] | None] = {<TopAbs_ShapeEnum.TopAbs_COMPOUND: 0>: <built-in method VolumeProperties_s of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_COMPSOLID: 1>: <built-in method VolumeProperties_s of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_EDGE: 6>: <built-in method LinearProperties_s of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_FACE: 4>: <built-in method SurfaceProperties_s of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_SHELL: 3>: <built-in method SurfaceProperties_s of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_SOLID: 2>: <built-in method VolumeProperties_s of PyCapsule object>, <TopAbs_ShapeEnum.TopAbs_VERTEX: 7>: None, <TopAbs_ShapeEnum.TopAbs_WIRE: 5>: <built-in method LinearProperties_s of PyCapsule object>}

property shape_type: Literal['Vertex', 'Edge', 'Wire', 'Face', 'Shell', 'Solid', 'Compound']

Return the shape type string for this class

shell() → *Shell*

Return the Shell

shells() → *ShapeList*[*Shell*]

shells - all the shells in this Shape

show_topology(*limit_class: Literal['Compound', 'Edge', 'Face', 'Shell', 'Solid', 'Vertex', 'Wire'] = 'Vertex'*, *show_center: bool | None = None*) → str

Display internal topology

Display the internal structure of a Compound 'assembly' or Shape. Example:

```
>>> c1.show_topology()

c1 is the root         Compound at 0x7f4a4cafafa0, Location(...))
├──                    Solid    at 0x7f4a4cafafd0, Location(...))
├── c2 is 1st compound Compound at 0x7f4a4cafaee0, Location(...))
│   ├──                Solid    at 0x7f4a4cafad00, Location(...))
│   └──                Solid    at 0x7f4a11a52790, Location(...))
└── c3 is 2nd          Compound at 0x7f4a4cafad60, Location(...))
    ├──                Solid    at 0x7f4a11a52700, Location(...))
    └──                Solid    at 0x7f4a11a58550, Location(...))
```

Parameters

- **limit_class** – type of displayed leaf node. Defaults to 'Vertex'.
- **show_center** (*bool, optional*) – If None, shows the Location of Compound 'assemblies' and the bounding box center of Shapes. True or False forces the display. Defaults to None.

Returns

tree representation of internal structure

Return type
str

solid() → *Solid*

Return the Solid

solids() → *ShapeList*[*Solid*]

solids - all the solids in this Shape

split(*tool: TrimmingTool*, *keep: Literal[Keep.TOP, Keep.BOTTOM]*) → Self | list[Self] | None

split(*tool: TrimmingTool*, *keep: Literal[Keep.ALL]*) → list[Self]

split(*tool: TrimmingTool*, *keep: Literal[Keep.BOTH]*) → tuple[Self | list[Self] | None, Self | list[Self] | None]

split(*tool: TrimmingTool*, *keep: Literal[Keep.INSIDE, Keep.OUTSIDE]*) → None

split(*tool: TrimmingTool*) → Self | list[Self] | None

split

Split this shape by the provided plane or face.

Parameters

- **surface** (Plane | Face) – surface to segment shape
- **keep** (Keep, *optional*) – which object(s) to save. Defaults to Keep.TOP.

Returns
result of split

Return type
Shape

Returns

Self | list[Self] | None, Tuple[Self | list[Self] | None]: The result of the split operation.

- **Keep.TOP**: Returns the top as a *Self* or *list[Self]*, or *None* if no top is found.
- **Keep.BOTTOM**: Returns the bottom as a *Self* or *list[Self]*, or *None* if no bottom is found.
- **Keep.BOTH**: Returns a tuple *(inside, outside)* where each element is either a *Self* or *list[Self]*, or *None* if no corresponding part is found.

split_by_perimeter(*perimeter:* Edge | Wire, *keep: Literal[Keep.INSIDE, Keep.OUTSIDE]*) → *Face* | *Shell* | *ShapeList*[*Face*] | None

split_by_perimeter(*perimeter:* Edge | Wire, *keep: Literal[Keep.BOTH]*) → tuple[*Face* | *Shell* | *ShapeList*[*Face*] | None, *Face* | *Shell* | *ShapeList*[*Face*] | None]

split_by_perimeter(*perimeter:* Edge | Wire, *keep: Literal[Keep.INSIDE] = Keep.INSIDE*) → *Face* | *Shell* | *ShapeList*[*Face*] | None

split_by_perimeter

Divide the faces of this object into those within the perimeter and those outside the perimeter.

Note: this method may fail if the perimeter intersects shape edges.

Parameters

- **perimeter** (*Union*[Edge, Wire]) – closed perimeter
- **keep** (Keep, *optional*) – which object(s) to return. Defaults to Keep.INSIDE.

Raises

- **ValueError** – perimeter must be closed

- **ValueError** – keep must be one of Keep.INSIDE|OUTSIDE|BOTH

Returns

Union[Face | Shell | ShapeList[Face] | None, Tuple[Face | Shell | ShapeList[Face] | None]: The result of the split operation.

- **Keep.INSIDE**: Returns the inside part as a *Shell* or *Face*, or *None* if no inside part is found.
- **Keep.OUTSIDE**: Returns the outside part as a *Shell* or *Face*, or *None* if no outside part is found.
- **Keep.BOTH**: Returns a tuple *(inside, outside)* where each element is either a *Shell*, *Face*, or *None* if no corresponding part is found.

property static_moments: tuple[float, float, float]

Compute the static moments (first moments of mass) of the shape.

The static moments represent the weighted sum of the coordinates with respect to the mass distribution, providing insight into the center of mass and mass distribution of the shape.

Returns

The static moments (Mx, My, Mz), where: - Mx is the first moment of mass about the YZ plane. - My is the first moment of mass about the XZ plane. - Mz is the first moment of mass about the XY plane.

Return type

tuple[*float*, *float*, *float*]

Example

```
>>> obj = MyShape()
>>> obj.static_moments
(150.0, 200.0, 50.0)
```

tessellate(*tolerance: float*, *angular_tolerance: float = 0.1*) → tuple[list[Vector], list[tuple[int, int, int]]]

General triangulated approximation

to_splines(*degree: int = 3*, *tolerance: float = 0.001*, *nurbs: bool = False*) → Self

A shape-processing utility that forces all geometry in a shape to be converted into BSplines. It's useful when working with tools or export formats that require uniform geometry, or for downstream processing that only understands BSpline representations.

Parameters

- **degree** (*int, optional*) – Maximum degree. Defaults to 3.
- **tolerance** (*float, optional*) – Approximation tolerance. Defaults to 1e-3.
- **nurbs** (*bool, optional*) – Use rational splines. Defaults to False.

Returns

Approximated shape

Return type

Self

touch(*other:* Shape, *tolerance: float = 1e-06*) → *ShapeList*

Find boundary contacts between this shape and another.

Base implementation returns empty ShapeList. Subclasses (Mixin2D, Mixin3D, Compound) override this to provide actual touch detection.

Parameters

- **other** – Shape to find contacts with
- **tolerance** – tolerance for contact detection

Returns
ShapeList of contact shapes (empty for base implementation)

transform_geometry(*t_matrix: Matrix*) → Self

Apply affine transform

WARNING: transform_geometry will sometimes convert lines and circles to splines, but it also has the ability to handle skew and stretching transformations.

If your transformation is only translation and rotation, it is safer to use *transform_shape()*, which doesn't change the underlying type of the geometry, but cannot handle skew transformations.

Parameters
t_matrix (Matrix) – affine transformation matrix

Returns
a copy of the object, but with geometry transformed

Return type
Shape

transform_shape(*t_matrix: Matrix*) → Self

Apply affine transform without changing type

Transforms a copy of this Shape by the provided 3D affine transformation matrix. Note that not all transformation are supported - primarily designed for translation and rotation. See :transform_geometry: for more comprehensive transformations.

Parameters
t_matrix (Matrix) – affine transformation matrix

Returns
copy of transformed shape with all objects keeping their type

Return type
Shape

transformed(*rotate: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 0)*, *offset: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] = (0, 0, 0)*) → Self

Transform Shape

Rotate and translate the Shape by the three angles (in degrees) and offset.

Parameters

- **rotate** (*VectorLike, optional*) – 3-tuple of angles to rotate, in degrees. Defaults to (0, 0, 0).
- **offset** (*VectorLike, optional*) – 3-tuple to offset. Defaults to (0, 0, 0).

Returns
transformed object

Return type
Shape

translate(*vector: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *transform: bool = False*) → Self

Translates this shape through a transformation.

Parameters

- **vector** (*VectorLike*) – relative movement vector
- **transform** (*bool*) – regenerate the shape instead of just changing its location Defaults to False.

Returns
object with a relative move applied

vertex() → *Vertex*

Return the Vertex

vertices() → *ShapeList*[*Vertex*]

vertices - all the vertices in this Shape

wire() → *Wire*

Return the Wire

wires() → *ShapeList*[*Wire*]

wires - all the wires in this Shape

property wrapped

OCP TopoDS object

class ShapeList(*iterable=()*, */*)

Subclass of list with custom filter and sort methods appropriate to CAD

__and__(*other:* ShapeList) → *ShapeList*[T]

Intersect two ShapeLists operator &

__getitem__(*key: SupportsIndex*) → T

__getitem__(*key: slice*) → *ShapeList*[T]

Return slices of ShapeList as ShapeList

__gt__(*sort_by: Axis | SortBy = Axis((0, 0, 0), (0, 0, 1))*) → *ShapeList*[T]

Sort operator >

__lshift__(*group_by: Axis | SortBy = Axis((0, 0, 0), (0, 0, 1))*) → *ShapeList*[T]

Group and select smallest group operator <<

__lt__(*sort_by: Axis | SortBy = Axis((0, 0, 0), (0, 0, 1))*) → *ShapeList*[T]

Reverse sort operator <

__or__(*filter_by: Axis | GeomType = Axis((0, 0, 0), (0, 0, 1))*) → *ShapeList*[T]

Filter by axis or geomtype operator |

__rshift__(*group_by: Axis | SortBy = Axis((0, 0, 0), (0, 0, 1))*) → *ShapeList*[T]

Group and select largest group operator >>

__sub__(*other:* ShapeList) → *ShapeList*[T]

Differences between two ShapeLists operator -

center() → Vector

The average of the center of objects within the ShapeList

compound() → *Compound*

Return the Compound

compounds() → *ShapeList*[*Compound*]

compounds - all the compounds in this ShapeList

edge() → *Edge*

Return the Edge

edges() → *ShapeList*[*Edge*]

edges - all the edges in this ShapeList

expand() → *ShapeList*

Expand by dissolving compounds, wires, and shells, filtering nulls.

Returns
ShapeList with compounds dissolved to children, wires to edges, shells to faces, and nulls filtered out

face() → *Face*

Return the Face

faces() → *ShapeList*[*Face*]

faces - all the faces in this ShapeList

filter_by(*filter_by: Callable[[T], bool] | Axis | Plane | GeomType | property*, *reverse: bool = False*, *tolerance: float = 1e-05*) → *ShapeList*[T]

filter by

Either: - filter objects of type planar Face or linear Edge by their normal or tangent (respectively) and sort the results by the given axis, or - filter the objects by the provided type. Note that not all types apply to all objects.

Parameters

- **filter_by** (*Callable[[T], bool] | Axis | Plane | GeomType*) – function, axis, plane, or geom type to filter and possibly sort by. Filtering by a plane returns faces/edges parallel to that plane.
- **reverse** (*bool, optional*) – invert the geom type filter. Defaults to False.
- **tolerance** (*float, optional*) – maximum deviation from axis. Defaults to 1e-5.

Raises
ValueError – Invalid filter_by type

Returns
filtered list of objects

Return type
ShapeList

filter_by_position(*axis: Axis*, *minimum: float*, *maximum: float*, *inclusive: tuple[bool, bool] = (True, True)*) → *ShapeList*[T]

filter by position

Filter and sort objects by the position of their centers along given axis. min and max values can be inclusive or exclusive depending on the inclusive tuple.

Parameters

- **axis** (*Axis*) – axis to sort by

- **minimum** (*float*) – minimum value
- **maximum** (*float*) – maximum value
- **inclusive** (*tuple[bool, bool], optional*) – include min,max values. Defaults to (True, True).

Returns
filtered object list

Return type
ShapeList

property first: T

First element in the ShapeList

group_by(*group_by: Callable[[T], K] | Axis | Edge | Wire | SortBy | property = Axis((0, 0, 0), (0, 0, 1))*, *reverse: bool = False*, *tol_digits: int = 6*) → GroupBy[T, K]

group by

Group objects by provided criteria and then sort the groups according to the criteria. Note that not all group_by criteria apply to all objects.

Parameters

- **(Callable[[T]** (*group_by*) – optional): group and sort criteria. Defaults to Axis.Z.
- **property** (*K] | Axis | Edge | Wire | SortBy |*) – optional): group and sort criteria. Defaults to Axis.Z.

:param : optional): group and sort criteria. Defaults to Axis.Z. :param reverse: flip order of sort. Defaults to False. :type reverse: bool, optional :param tol_digits: Tolerance for building the group keys by

round(key, tol_digits)

Returns
sorted groups of ShapeLists

Return type
GroupBy[T, K]

property last: T

Last element in the ShapeList

shell() → *Shell*

Return the Shell

shells() → *ShapeList*[*Shell*]

shells - all the shells in this ShapeList

solid() → *Solid*

Return the Solid

solids() → *ShapeList*[*Solid*]

solids - all the solids in this ShapeList

sort_by(*sort_by: Callable[[T], K] | Axis | Edge | Wire | SortBy | property = Axis((0, 0, 0), (0, 0, 1))*, *reverse: bool = False*) → *ShapeList*[T]

sort by

Sort objects by provided criteria. Note that not all sort_by criteria apply to all objects.

Parameters

- **(Callable[[T]** (*sort_by*) – optional): sort criteria. Defaults to Axis.Z.
- **property** (*K] | Axis | Edge | Wire | SortBy |*) – optional): sort criteria. Defaults to Axis.Z.

:param : optional): sort criteria. Defaults to Axis.Z. :param reverse: flip order of sort. Defaults to False. :type reverse: bool, optional

Raises

- **ValueError** – Cannot sort by an empty axis
- **ValueError** – Cannot sort by an empty object
- **ValueError** – Invalid sort_by criteria provided

Returns
sorted list of objects

Return type
ShapeList

sort_by_distance(*other:* Shape | *Vector* | *tuple[float, float]* | *tuple[float, float, float]* | *Sequence[float]*, *reverse: bool = False*) → *ShapeList*[T]

Sort by distance

Sort by minimal distance between objects and other

Parameters

- **other** (*Union[Shape,VectorLike]*) – reference object
- **reverse** (*bool, optional*) – flip order of sort. Defaults to False.

Returns
Sorted shapes

Return type
ShapeList

vertex() → *Vertex*

Return the Vertex

vertices() → *ShapeList*[*Vertex*]

vertices - all the vertices in this ShapeList

wire() → *Wire*

Return the Wire

wires() → *ShapeList*[*Wire*]

wires - all the wires in this ShapeList

class Shell(*obj: TopoDS_Shell* | Face | *Iterable[Face]* | *None = None*, *label: str = ''*, *color:* Color | *None = None*, *parent:* Compound | *None = None*)

A Shell is a fundamental component in build123d's topological data structure representing a connected set of faces forming a closed surface in 3D space. As part of a geometric model, it defines a watertight enclosure, commonly encountered in solid modeling. Shells group faces in a coherent manner, playing a crucial role in representing complex shapes with voids and surfaces. This hierarchical structure allows for efficient handling of surfaces within a model, supporting various operations and analyses.

center() → Vector

Center of mass of the shell

classmethod extrude(*obj:* Wire, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Shell*

Extrude a Wire into a Shell.

Parameters
direction (*VectorLike*) – direction and magnitude of extrusion

Raises
- **ValueError** – Unsupported class
- **RuntimeError** – Generated invalid result

Returns
extruded shape

Return type
Edge

location_at(*surface_point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, ***, *x_dir: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → Location

Get the location (origin and orientation) on the surface of the shell.

Parameters
- **surface_point** (*VectorLike*) – A 3D point near the surface.
- **x_dir** (*VectorLike, optional*) – Direction for the local X axis. If not given, the tangent in the U direction is used.

Returns
A full 3D placement at the specified point on the shell surface.

Return type
Location

classmethod make_loft(*objs: Iterable[*Vertex | Wire*]*, *ruled: bool = False*) → *Shell*

make loft

Makes a loft from a list of wires and vertices. Vertices can appear only at the beginning or end of the list, but cannot appear consecutively within the list nor between wires. Wires may be closed or opened.

Parameters
- **objs** (*list[*Vertex, Wire*]*) – wire perimeters or vertices
- **ruled** (*bool, optional*) – stepped or smooth. Defaults to False (smooth).

Raises
ValueError – Too few wires

Returns
Lofted object

Return type
Shell

order = 2.5

classmethod revolve(*profile:* Curve | Wire, *angle: float*, *axis:* Axis) → *Face*

sweep

Revolve a 1D profile around an axis.

Parameters

- **profile** (Curve | Wire) – the object to revolve
- **angle** (*float*) – the angle to revolve through
- **axis** (Axis) – rotation Axis

Returns
resulting shell

Return type
Shell

classmethod sweep(*profile: Curve | Edge | Wire*, *path: Curve | Edge | Wire*, *transition=<Transition.TRANSFORMED>*) → *Shell*

Sweep a 1D profile along a 1D path

Parameters

- **profile** (*Union[Curve, Edge, Wire]*) – the object to sweep
- **path** (*Union[Curve, Edge, Wire]*) – the path to follow when sweeping
- **transition** (Transition, *optional*) – handling of profile orientation at C1 path discontinuities. Defaults to Transition.TRANSFORMED.

Returns
resulting Shell, may be non-planar

Return type
Shell

property volume: float

volume - the volume of this Shell if manifold, otherwise zero

class Solid(*obj: TopoDS_Solid | Shell | None = None*, *label: str = ''*, *color:* Color | *None = None*, *material: str = ''*, *joints: dict[str,* Joint*] | None = None*, *parent:* Compound | *None = None*)

A Solid in build123d represents a three-dimensional solid geometry in a topological structure. A solid is a closed and bounded volume, enclosing a region in 3D space. It comprises faces, edges, and vertices connected in a well-defined manner. Solid modeling operations, such as Boolean operations (union, intersection, and difference), are often performed on Solid objects to create or modify complex geometries.

draft(*faces: Iterable[*Face*]*, *neutral_plane: Plane*, *angle: float*) → *Solid*

Apply a draft angle to the given faces of the solid.

Parameters

- **faces** – Faces to which the draft should be applied.
- **neutral_plane** – Plane defining the neutral direction and position.
- **angle** – Draft angle in degrees.

Returns
Solid with the specified draft angles applied.

Raises
RuntimeError – If draft application fails on any face or during build.

classmethod extrude(*obj: Face*, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Solid*

Extrude a Face into a Solid.

Parameters
direction (*VectorLike*) – direction and magnitude of extrusion

Raises

- **ValueError** – Unsupported class
- **RuntimeError** – Generated invalid result

Returns
extruded shape

Return type
Edge

classmethod extrude_linear_with_rotation(*section: Face | Wire*, *center: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *normal: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *angle: float*, *inner_wires: list[Wire] | None = None*) → *Solid*

Extrude with Rotation

Creates a 'twisted prism' by extruding, while simultaneously rotating around the extrusion vector.

Parameters

- **section** (*Union[Face,Wire]*) – cross section
- **vec_center** (*VectorLike*) – the center point about which to rotate
- **vec_normal** (*VectorLike*) – a vector along which to extrude the wires
- **angle** (*float*) – the angle to rotate through while extruding
- **inner_wires** (*list[Wire], optional*) – holes - only used if section is of type Wire. Defaults to None.

Returns
extruded object

Return type
Solid

classmethod extrude_taper(*profile: Face*, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *taper: float*, *flip_inner: bool = True*) → *Solid*

Extrude a cross section with a taper

Extrude a cross section into a prismatic solid in the provided direction.

Note that two difference algorithms are used. If direction aligns with the profile normal (which must be positive), the taper is positive and the profile contains no holes the OCP LocOpe_DPrism algorithm is used as it generates the most accurate results. Otherwise, a loft is created between the profile and the profile with a 2D offset set at the appropriate direction.

Parameters

- **section** (*Face]*) – cross section
- **normal** (*VectorLike*) – a vector along which to extrude the wires. The length of the vector controls the length of the extrusion.

- **taper** (*float*) – taper angle in degrees.
- **flip_inner** (*bool, optional*) – outer and inner geometry have opposite tapers to allow for part extraction when injection molding.

Returns
extruded cross section

Return type
Solid

classmethod extrude_until(*profile: Face*, *target: Compound | Solid*, *direction: VectorLike*, *until: Until = <Until.NEXT>*) → *Solid*

Extrude *profile* in the provided *direction* until it encounters a bounding surface on the *target*. The termination surface is chosen according to the *until* option:

- `Until.NEXT` — Extrude forward until the first intersecting surface.
- `Until.LAST` — Extrude forward through all intersections, stopping at

the farthest surface. * `Until.PREVIOUS` — Reverse the extrusion direction and stop at the first intersecting surface behind the profile. * `Until.FIRST` — Reverse the direction and stop at the farthest surface behind the profile.

When `Until.PREVIOUS` or `Until.FIRST` are used, the extrusion direction is automatically inverted before execution.

> **Note**
>
> The bounding surface on the target must be large enough to completely cover the extruded profile at the contact region. Partial overlaps may yield open or invalid solids.

Parameters

- **profile** (`Face`) – The face to extrude.
- **target** (*Union[Compound, Solid]*) – The object that limits the extrusion.
- **direction** (*VectorLike*) – Extrusion direction.
- **until** (`Until`, *optional*) – Surface selection mode controlling which intersection to stop at. Defaults to `Until.NEXT`.

Raises
ValueError – If the provided profile does not intersect the target.

Returns
The extruded and limited solid.

Return type
Solid

classmethod from_bounding_box(*bbox: BoundBox | OrientedBoundBox*) → *Solid*

A box of the same dimensions and location

classmethod make_box(*length: float*, *width: float*, *height: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Solid*

make box

Make a box at the origin of plane extending in positive direction of each axis.

Parameters

- **length** (*float*)
- **width** (*float*)
- **height** (*float*)
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.

Returns
Box

Return type
Solid

classmethod make_cone(*base_radius: float*, *top_radius: float*, *height: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *angle: float = 360*) → *Solid*

make cone

Make a cone with given radii and height

Parameters

- **base_radius** (*float*)
- **top_radius** (*float*)
- **height** (*float*)
- **plane** (Plane) – base plane. Defaults to Plane.XY.
- **angle** (*float, optional*) – arc size. Defaults to 360.

Returns
Full or partial cone

Return type
Solid

classmethod make_cylinder(*radius: float*, *height: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *angle: float = 360*) → *Solid*

make cylinder

Make a cylinder with a given radius and height with the base center on plane origin.

Parameters

- **radius** (*float*)
- **height** (*float*)
- **plane** (Plane) – base plane. Defaults to Plane.XY.
- **angle** (*float, optional*) – arc size. Defaults to 360.

Returns
Full or partial cylinder

Return type
Solid

classmethod make_loft(*objs: Iterable[*Vertex | Wire*]*, *ruled: bool = False*) → *Solid*

make loft

Makes a loft from a list of wires and vertices. Vertices can appear only at the beginning or end of the list, but cannot appear consecutively within the list nor between wires.

Parameters

- **objs** (*list[Vertex, Wire]*) – wire perimeters or vertices
- **ruled** (*bool, optional*) – stepped or smooth. Defaults to False (smooth).

Raises
ValueError – Too few wires

Returns
Lofted object

Return type
Solid

classmethod make_sphere(*radius: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *angle1: float = -90*, *angle2: float = 90*, *angle3: float = 360*) → *Solid*

Sphere

Make a full or partial sphere - with a given radius center on the origin or plane.

Parameters

- **radius** (*float*)
- **plane** (Plane) – base plane. Defaults to Plane.XY.
- **angle1** (*float, optional*) – Defaults to -90.
- **angle2** (*float, optional*) – Defaults to 90.
- **angle3** (*float, optional*) – Defaults to 360.

Returns
sphere

Return type
Solid

classmethod make_torus(*major_radius: float*, *minor_radius: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *start_angle: float = 0*, *end_angle: float = 360*, *major_angle: float = 360*) → *Solid*

make torus

Make a torus with a given radii and angles

Parameters

- **major_radius** (*float*)
- **minor_radius** (*float*)
- **plane** (Plane) – base plane. Defaults to Plane.XY.
- **start_angle** (*float, optional*) – start major arc. Defaults to 0.
- **end_angle** (*float, optional*) – end major arc. Defaults to 360.

Returns
Full or partial torus

Return type
Solid

classmethod make_wedge(*delta_x: float*, *delta_y: float*, *delta_z: float*, *min_x: float*, *min_z: float*, *max_x: float*, *max_z: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Solid*

Make a wedge

Parameters

- **delta_x** (*float*)
- **delta_y** (*float*)
- **delta_z** (*float*)
- **min_x** (*float*)
- **min_z** (*float*)
- **max_x** (*float*)
- **max_z** (*float*)
- **plane** (Plane) – base plane. Defaults to Plane.XY.

Returns
wedge

Return type
Solid

order = 3.0

classmethod revolve(*section: Face | Wire*, *angle: float*, *axis: Axis*, *inner_wires: list[Wire] | None = None*) → *Solid*

Revolve

Revolve a cross section about the given Axis by the given angle.

Parameters

- **section** (*Union[Face, Wire]*) – cross section
- **angle** (*float*) – the angle to revolve through
- **axis** (Axis) – rotation Axis
- **inner_wires** (*list[Wire], optional*) – holes - only used if section is of type Wire. Defaults to [].

Returns
the revolved cross section

Return type
Solid

classmethod sweep(*section: ~topology.two_d.Face | ~topology.one_d.Wire*, *path: ~topology.one_d.Wire | ~topology.one_d.Edge*, *inner_wires: list[~topology.one_d.Wire] | None = None*, *make_solid: bool = True*, *is_frenet: bool = False*, *mode: ~build123d.geometry.Vector | ~topology.one_d.Wire | ~topology.one_d.Edge | None = None*, *transition: ~build123d.build_enums.Transition = <Transition.TRANSFORMED>*) → *Solid*

Sweep

Sweep the given cross section into a prismatic solid along the provided path

The is_frenet parameter controls how the profile orientation changes as it follows along the sweep path. If is_frenet is False, the orientation of the profile is kept consistent from point to point. The resulting shape has the minimum possible twisting. Unintuitively, when a profile is swept along a helix, this results in the

orientation of the profile slowly creeping (rotating) as it follows the helix. Setting is_frenet to True prevents this.

If is_frenet is True the orientation of the profile is based on the local curvature and tangency vectors of the path. This keeps the orientation of the profile consistent when sweeping along a helix (because the curvature vector of a straight helix always points to its axis). However, when path is not a helix, the resulting shape can have strange looking twists sometimes. For more information, see Frenet Serret formulas http://en.wikipedia.org/wiki/Frenet%E2%80%93Serret_formulas.

Parameters

- **section** (*Union[Face, Wire]*) – cross section to sweep
- **path** (*Union[Wire, Edge]*) – sweep path
- **inner_wires** (*list[Wire]*) – holes - only used if section is a wire
- **make_solid** (*bool, optional*) – return Solid or Shell. Defaults to True.
- **is_frenet** (*bool, optional*) – Frenet mode. Defaults to False.
- **mode** (*Union[Vector, Wire, Edge, None], optional*) – additional sweep mode parameters. Defaults to None.
- **transition** (*Transition, optional*) – handling of profile orientation at C1 path discontinuities. Defaults to Transition.TRANSFORMED.

Returns
the swept cross section

Return type
Solid

classmethod sweep_multi(*profiles: Iterable[Wire | Face]*, *path: Wire | Edge*, *make_solid: bool = True*, *is_frenet: bool = False*, *binormal: Vector | Wire | Edge | None = None*) → *Solid*

Multi section sweep

Sweep through a sequence of profiles following a path.

The is_frenet parameter controls how the profile orientation changes as it follows along the sweep path. If is_frenet is False, the orientation of the profile is kept consistent from point to point. The resulting shape has the minimum possible twisting. Unintuitively, when a profile is swept along a helix, this results in the orientation of the profile slowly creeping (rotating) as it follows the helix. Setting is_frenet to True prevents this.

If is_frenet is True the orientation of the profile is based on the local curvature and tangency vectors of the path. This keeps the orientation of the profile consistent when sweeping along a helix (because the curvature vector of a straight helix always points to its axis). However, when path is not a helix, the resulting shape can have strange looking twists sometimes. For more information, see Frenet Serret formulas http://en.wikipedia.org/wiki/Frenet%E2%80%93Serret_formulas.

Parameters

- **profiles** (*Iterable[Union[Wire, Face]]*) – list of profiles
- **path** (*Union[Wire, Edge]*) – The wire to sweep the face resulting from the wires over
- **make_solid** (*bool, optional*) – Solid or Shell. Defaults to True.
- **is_frenet** (*bool, optional*) – Select frenet mode. Defaults to False.
- **binormal** (*Union[Vector, Wire, Edge, None], optional*) – additional sweep mode parameters. Defaults to None.

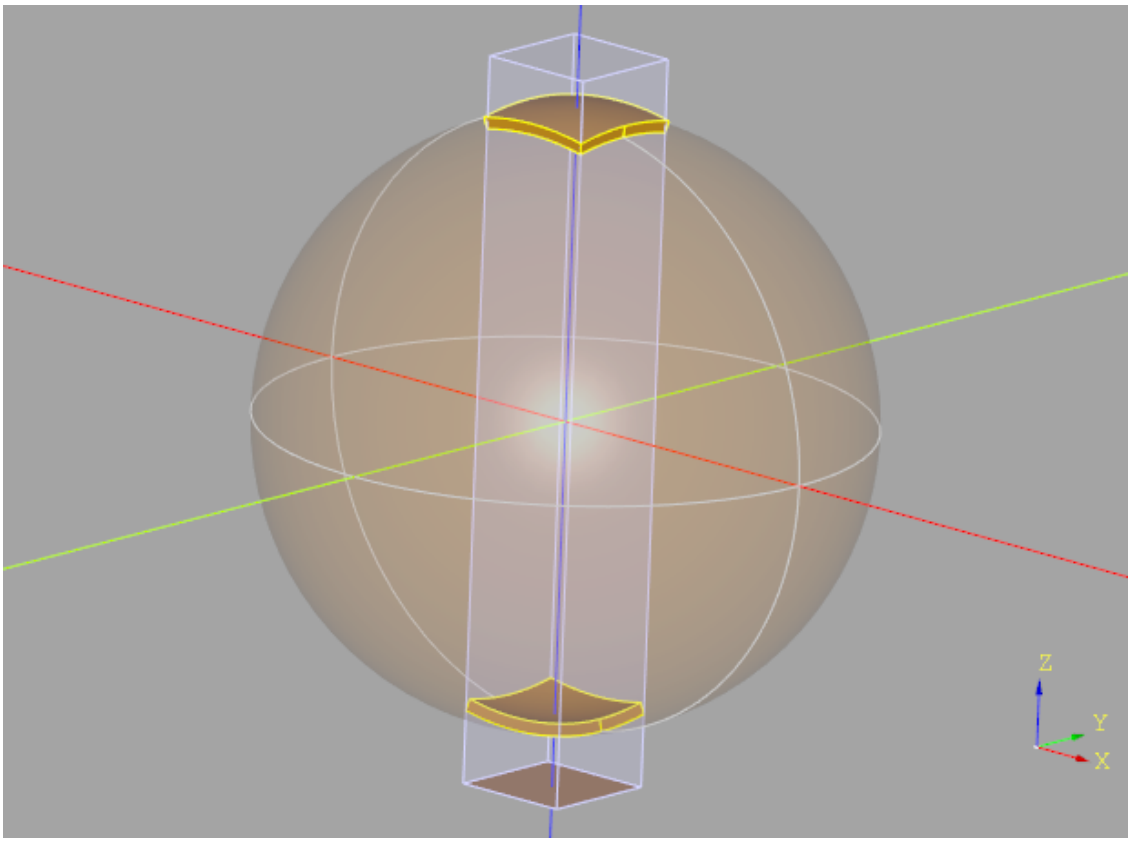
Returns
swept object

Return type
Solid

classmethod thicken(*surface:* Face | Shell, *depth: float*, *normal_override: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → *Solid*

Thicken Face or Shell

Create a solid from a potentially non planar face or shell by thickening along the normals.



Non-planar faces are thickened both towards and away from the center of the sphere.

Parameters

- **depth** (*float*) – Amount to thicken face(s), can be positive or negative.
- **normal_override** (Vector, *optional*) – Face only. The normal_override vector can be used to indicate which way is 'up', potentially flipping the face normal direction such that many faces with different normals all go in the same direction (direction need only be +/- 90 degrees from the face normal). Defaults to None.

Raises
RuntimeError – Opencascade internal failures

Returns
The resulting Solid object

Return type
Solid

touch(*other:* Shape, *tolerance: float = 1e-06*, *found_solids:* ShapeList | *None = None*) → *ShapeList*[*Vertex* | *Edge* | *Face*]

Find where this Solid's boundary contacts another shape.

Returns geometry where boundaries contact without interior overlap: - Solid + Solid → Face + Edge + Vertex (all boundary contacts) - Solid + Face/Shell → Face + Edge + Vertex (boundary contacts) - Solid + Edge/Wire → Vertex (edge endpoints on solid boundary) - Solid + Vertex → Vertex if on boundary - Solid + Compound → distributes over compound elements

Parameters

- **other** – Shape to check boundary contacts with
- **tolerance** – tolerance for contact detection
- **found_solids** – pre-found intersection solids to filter against

Returns
ShapeList of boundary contact geometry (empty if no contact)

property volume: float

volume - the volume of this Solid

class Wire(*obj: TopoDS_Wire*, *label: str = ''*, *color: Color | None = None*, *parent:* Compound | *None = None*)

class Wire(*edge:* Edge, *label: str = ''*, *color: Color | None = None*, *parent:* Compound | *None = None*)

class Wire(*wire:* Wire, *label: str = ''*, *color: Color | None = None*, *parent:* Compound | *None = None*)

class Wire(*wire:* Curve, *label: str = ''*, *color: Color | None = None*, *parent:* Compound | *None = None*)

class Wire(*edges: Iterable[*Edge*]*, *sequenced: bool = False*, *label: str = ''*, *color: Color | None = None*, *parent:* Compound | *None = None*)

A Wire in build123d is a topological entity representing a connected sequence of edges forming a continuous curve or path in 3D space. Wires are essential components in modeling complex objects, defining boundaries for surfaces or solids. They store information about the connectivity and order of edges, allowing precise definition of paths within a 3D model.

chamfer_2d(*distance: float*, *distance2: float*, *vertices: Iterable[*Vertex*]*, *edge:* Edge | *None = None*) → *Wire*

Apply 2D chamfer to a wire

Parameters

- **distance** (*float*) – chamfer length
- **distance2** (*float*) – chamfer length
- **vertices** (*Iterable[Vertex]*) – vertices to chamfer
- **edge** (Edge) – identifies the side where length is measured. The vertices must be part of the edge

Returns
chamfered wire

Return type
Wire

close() → *Wire*

Close a Wire

classmethod combine(*wires: Iterable[*Wire | Edge*]*, *tol: float = 1e-09*) → *ShapeList*[*Wire*]

Combine a list of wires and edges into a list of Wires.

Parameters

- **wires** (*Iterable[Wire | Edge]*) – unsorted
- **tol** (*float, optional*) – tolerance. Defaults to 1e-9.

Returns
Wires

Return type
ShapeList[Wire]

edges() → *ShapeList*[*Edge*]

edges - all the edges in this Shape

classmethod extrude(*obj:* Shape, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Wire*

extrude - invalid operation for Wire

fillet_2d(*radius: float*, *vertices: Iterable[*Vertex*]*) → *Wire*

Apply 2D fillet to a wire

Parameters

- **radius** (*float*)
- **vertices** (*Iterable[Vertex]*) – vertices to fillet

Raises

- **RuntimeError** – Internal error
- **ValueError** – empty wire

Returns
filleted wire

Return type
Wire

fix_degenerate_edges(*precision: float*) → *Wire*

Fix a Wire that contains degenerate (very small) edges

Parameters
precision (*float*) – minimum value edge length

Returns
fixed wire

Return type
Wire

geom_adaptor() → BRepAdaptor_CompCurve

Return the Geom Comp Curve for this Wire

geom_equal(*other:* Wire, *tol: float = 1e-06*, *num_interpolation_points: int = 5*) → bool

Compare two wires for geometric equality within tolerance.

This compares the geometric properties of two wires by comparing their constituent edges pairwise. Two independently created wires with the same geometry will return True.

Parameters

- **other** – Wire to compare with
- **tol** – Tolerance for numeric comparisons. Defaults to 1e-6.

- **num_interpolation_points** – Number of points to sample for unknown curve types. Defaults to 5.

Returns
True if wires are geometrically equal within tolerance

Return type
bool

classmethod make_circle(*radius: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Wire*

Makes a circle centered at the origin of plane

Parameters

- **radius** (*float*) – circle radius
- **plane** (Plane) – base plane. Defaults to Plane.XY

Returns
a circle

Return type
Wire

classmethod make_convex_hull(*edges: Iterable[*Edge*]*, *tolerance: float = 0.001*) → *Wire*

Create a wire of minimum length enclosing all of the provided edges.

Note that edges can't overlap each other.

Parameters

- **edges** (*Iterable[*Edge*]*) – edges defining the convex hull
- **tolerance** (*float*) – allowable error as a fraction of each edge length. Defaults to 1e-3.

Raises
ValueError – edges overlap

Returns
convex hull perimeter

Return type
Wire

classmethod make_ellipse(*x_radius: float*, *y_radius: float*, *plane: ~build123d.geometry.Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*, *start_angle: float = 360.0*, *end_angle: float = 360.0*, *angular_direction: ~build123d.build_enums.AngularDirection = <AngularDirection.COUNTER_CLOCKWISE>*, *closed: bool = True*) → *Wire*

make ellipse

Makes an ellipse centered at the origin of plane.

Parameters

- **x_radius** (*float*) – x radius of the ellipse (along the x-axis of plane)
- **y_radius** (*float*) – y radius of the ellipse (along the y-axis of plane)
- **plane** (Plane, *optional*) – base plane. Defaults to Plane.XY.
- **start_angle** (*float, optional*) – _description_. Defaults to 360.0.
- **end_angle** (*float, optional*) – _description_. Defaults to 360.0.

- **angular_direction** (*AngularDirection, optional*) – arc direction. Defaults to AngularDirection.COUNTER_CLOCKWISE.
- **closed** (*bool, optional*) – close the arc. Defaults to True.

Returns
an ellipse

Return type
Wire

classmethod make_polygon(*vertices: Iterable[Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]]*, *close: bool = True*) → *Wire*

Create an irregular polygon by defining vertices

Parameters

- **vertices** (*Iterable[VectorLike]*)
- **close** (*bool, optional*) – close the polygon. Defaults to True.

Returns
an irregular polygon

Return type
Wire

classmethod make_rect(*width: float*, *height: float*, *plane: Plane = Plane((0, 0, 0), (1, 0, 0), (0, 0, 1))*) → *Wire*

Make Rectangle

Make a Rectangle centered on center with the given normal

Parameters

- **width** (*float*) – width (local x)
- **height** (*float*) – height (local y)
- **plane** (Plane, *optional*) – plane containing rectangle. Defaults to Plane.XY.

Returns
The centered rectangle

Return type
Wire

order = 1.5

static order_chamfer_edges(*reference_edge:* Edge | *None*, *edges: tuple[*Edge, Edge*]*) → tuple[*Edge*, *Edge*]

Order the edges of a chamfer relative to a reference Edge

order_edges() → *ShapeList*[*Edge*]

Return the edges in self ordered by wire direction and orientation

param_at(*position: float*) → float

Return the OCCT comp-curve parameter corresponding to the given wire position. This is *not* the edge composite parameter; it is the parameter of the wire's BRepAdaptor_CompCurve.

param_at_point(*point: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → float

Return the normalized wire parameter for the point closest to this wire.

This method projects the given point onto the wire, finds the nearest edge, and accumulates arc lengths to determine the fractional position along the entire wire. The result is normalized to the interval [0.0, 1.0], where:

- 0.0 corresponds to the start of the wire
- 1.0 corresponds to the end of the wire

Unlike the edge version of this method, the returned value is **not** an OCCT curve parameter, but a normalized parameter across the wire as a whole.

Parameters
point (*VectorLike*) – The point to project onto the wire.

Raises
ValueError – Can't find point on empty wire

Returns
Normalized parameter in [0.0, 1.0] representing the relative position of the projected point along the wire.

Return type
float

project_to_shape(*target_object: Shape, direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None, center: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float] | None = None*) → *ShapeList*[*Wire*]

Project Wire

Project a Wire onto a Shape generating new wires on the surfaces of the object one and only one of *direction* or *center* must be provided. Note that one or more wires may be generated depending on the topology of the target object and location/direction of projection.

To avoid flipping the normal of a face built with the projected wire the orientation of the output wires are forced to be the same as self.

Parameters

- **target_object** – Object to project onto
- **direction** – Parallel projection direction. Defaults to None.
- **center** – Conical center of projection. Defaults to None.
- **target_object** – Shape:
- **direction** – VectorLike: (Default value = None)
- **center** – VectorLike: (Default value = None)

Returns
Projected wire(s)

Raises
ValueError – Only one of direction or center must be provided

stitch(*other: Wire*) → *Wire*

Attempt to stitch wires

Parameters
other (*Wire*) – wire to combine

Raises
ValueError – Can't stitch empty wires

Returns
stitched wires

Return type
Wire

to_wire() → *Wire*

Return Wire - used as a pair with Edge.to_wire when self is Wire | Edge

trim(*start: float | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*, *end: float | Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Wire*

Trim a wire between [start, end] normalized over total length.

Parameters

- **start** (*float | VectorLike*) – normalized start position (0.0 to <1.0) or point
- **end** (*float | VectorLike*) – normalized end position (>0.0 to 1.0) or point

Returns
trimmed Wire

Return type
Wire

class Vertex

class Vertex(*ocp_vx: TopoDS_Vertex*)

class Vertex(*X: float*, *Y: float*, *Z: float*)

class Vertex(*v: Iterable[float]*)

A Vertex in build123d represents a zero-dimensional point in the topological data structure. It marks the endpoints of edges within a 3D model, defining precise locations in space. Vertices play a crucial role in defining the geometry of objects and the connectivity between edges, facilitating accurate representation and manipulation of 3D shapes. They hold coordinate information and are essential for constructing complex structures like wires, faces, and solids.

__add__(*other:* Vertex | *Vector* | *tuple[float, float, float]*) → *Vertex*

Add

Add to a Vertex with a Vertex, Vector or Tuple

Parameters
other – Value to add

Raises
TypeError – other not in [Tuple,Vector,Vertex]

Returns
Result

Example

part.faces(">z").vertices("<y and <x").val() + (0, 0, 15)

which creates a new Vertex 15 above one extracted from a part. One can add or subtract a *Vertex* , *Vector* or *tuple* of float values to a Vertex.

__sub__(*other:* Vertex | *Vector* | *tuple*) → *Vertex*

Subtract

Subtract a Vertex with a Vertex, Vector or Tuple from self

Parameters
other – Value to add

Raises
TypeError – other not in [Tuple,Vector,Vertex]

Returns
Result

Example

part.faces(">z").vertices("<y and <x").val() - Vector(10, 0, 0)

classmethod cast(*obj: TopoDS_Shape*) → Self

Returns the right type of wrapper, given a OCCT object

center() → Vector

The center of a vertex is itself!

classmethod extrude(*obj:* Shape, *direction: Vector | tuple[float, float] | tuple[float, float, float] | Sequence[float]*) → *Vertex*

extrude - invalid operation for Vertex

order = 0.0

split(*tool: TrimmingTool*, *keep: Keep = <Keep.TOP>*)

split - not implemented

to_tuple() → tuple[float, float, float]

Return vertex as three tuple of floats

transform_shape(*t_matrix: Matrix*) → *Vertex*

Apply affine transform without changing type

Transforms a copy of this Vertex by the provided 3D affine transformation matrix. Note that not all transformation are supported - primarily designed for translation and rotation. See :transform_geometry: for more comprehensive transformations.

Parameters
t_matrix (Matrix) – affine transformation matrix

Returns
copy of transformed shape with all objects keeping their type

Return type
Vertex

vertex() → *Vertex*

Return the Vertex

vertices() → *ShapeList*[*Vertex*]

vertices - all the vertices in this Shape

property volume: float

volume - the volume of this Vertex, which is always zero

class Curve(*obj: TopoDS_Compound | Iterable[Shape] | None = None, label: str = '', color: Color | None = None, material: str = '', joints: dict[str, Joint] | None = None, parent: Compound | None = None, children: Sequence[Shape] | None = None*)

A Compound containing 1D objects - aka Edges

__matmul__(*position: float*) → Vector

Position on curve operator @ - only works if continuous

__mod__(*position: float*) → Vector

Tangent on wire operator % - only works if continuous

wires() → *ShapeList*[*Wire*]

A list of wires created from the edges

class Part(*obj: TopoDS_Compound | Iterable[Shape] | None = None, label: str = '', color: Color | None = None, material: str = '', joints: dict[str, Joint] | None = None, parent: Compound | None = None, children: Sequence[Shape] | None = None*)

A Compound containing 3D objects - aka Solids

class Sketch(*obj: TopoDS_Compound | Iterable[Shape] | None = None, label: str = '', color: Color | None = None, material: str = '', joints: dict[str, Joint] | None = None, parent: Compound | None = None, children: Sequence[Shape] | None = None*)

A Compound containing 2D objects - aka Faces

1.22.3 Import/Export

Methods and functions specific to exporting and importing build123d objects are defined below.

import_brep(*file_name: PathLike | str | bytes*) → Shape

Import shape from a BREP file

Parameters
file_name (*Union[PathLike, str, bytes]*) – brep file

Raises
ValueError – file not found

Returns
build123d object

Return type
Shape

import_step(*filename: PathLike | str | bytes*) → Compound

Extract shapes from a STEP file and return them as a Compound object.

Parameters
file_name (*Union[PathLike, str, bytes]*) – file path of STEP file to import

Raises
ValueError – can't open file

Returns
contents of STEP file

Return type
Compound

import_stl(*file_name: ~os.PathLike | str | bytes*, *model_unit: ~build123d.build_enums.Unit = <Unit.MM>*) → Face

Extract shape from an STL file and return it as a Face reference object.

Note that importing with this method and creating a reference is very fast while creating an editable model (with Mesher) may take minutes depending on the size of the STL file.

Parameters

- **file_name** (*Union[PathLike, str, bytes]*) – file path of STL file to import
- **model_unit** (*Unit, optional*) – the default unit used when creating the model. For example, Blender defaults to Unit.M. Defaults to Unit.MM.

Raises

- **ValueError** – Could not import file
- **ValueError** – Invalid model_unit

Returns
STL model

Return type
Face

import_svg(*svg_file: str | Path | TextIO*, ***, *flip_y: bool = True*, *align: Align | tuple[Align, Align] | None = Align.MIN*, *ignore_visibility: bool = False*, *label_by: Literal['id', 'class', 'inkscape:label'] | str = 'id'*) → ShapeList[Wire | Face]

import_svg(*svg_file: str | Path | TextIO*, ***, *flip_y: bool = True*, *align: Align | tuple[Align, Align] | None = Align.MIN*, *ignore_visibility: bool = False*, *label_by: Literal['id', 'class', 'inkscape:label'] | str = 'id'*, *is_inkscape_label: bool | None = None*) → ShapeList[Wire | Face]

import_svg

Parameters

- **svg_file** (*Union[str, Path, TextIO]*) – svg file
- **flip_y** (*bool, optional*) – flip objects to compensate for svg orientation. Defaults to True.
- **align** (*Align | tuple[Align, Align] | None, optional*) – alignment of the SVG's viewbox, if None, the viewbox's origin will be at *(0,0,0)*. Defaults to Align.MIN.
- **ignore_visibility** (*bool, optional*) – Defaults to False.
- **label_by** (*str, optional*) – XML attribute to use for imported shapes' *label* property. Defaults to "id". Use *inkscape:label* to read labels set from Inkscape's "Layers and Objects" panel.

Raises
ValueError – unexpected shape type

Returns
objects contained in svg

Return type
ShapeList[*Union*[Wire, Face]]

import_svg_as_buildline_code(*file_name: PathLike | str | bytes*, *precision: int = 6*) → tuple[str, str]

translate_to_buildline_code

Translate the contents of the given svg file into executable build123d/BuildLine code.

Parameters

- **file_name** (*PathLike | str | bytes*) – svg file name
- **precision** (*int*) – # digits to round values to. Defaults to # digits in TOLERANCE

Returns
code, builder instance name

Return type
tuple[*str*, *str*]

1.22.4 Joint Object

Base Joint class which is used to position Solid and Compound objects relative to each other are defined below. The *Joints* section contains the class description of the derived Joint classes.

class Joint(*label: str*, *parent:* BuildPart | Solid | Compound)

Abstract Base Joint class - used to join two components together

Parameters
parent (*Union[Solid, Compound]*) – object that joint to bound to

Variables

- **label** (*str*) – user assigned label
- **parent** (Shape) – object joint is bound to
- **connected_to** (Joint) – joint that is connect to this joint

abstract connect_to(**args*, ***kwargs*)

All derived classes must provide a connect_to method

abstract property location: Location

Location of joint

abstract relative_to(**args*, ***kwargs*) → Location

Return relative location to another joint

abstract property symbol: *Compound*

A CAD object positioned in global space to illustrate the joint

1.23 Indices and tables

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PYTHON MODULE INDEX

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