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PyAnsys Geometry



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CONTENTS



PyAnsys Geometry

PyAnsys Geometry is a Python client library for the Ansys Geometry service. You are looking at the documentation for version 0.9.0.

Getting started Learn how to run the Windows Docker container, install the PyAnsys Geometry image, and launch and connect to the Geometry service.

Getting started **User guide** Understand key concepts and approaches for primitives, sketches, and model designs.

User guide **API reference** Understand PyAnsys Geometry API endpoints, their capabilities, and how to interact with them programmatically.

API reference **Examples** Explore examples that show how to use PyAnsys Geometry to perform many different types of operations.

Examples **Contribute** Learn how to contribute to the PyAnsys Geometry codebase or documentation.

Contribute **Assets** Download different assets related to PyAnsys Geometry, such as documentation, package wheelhouse, and related files.

Assets

PyAnsys Geometry is a Python client library for the Ansys Geometry service.

INSTALLATION

You can use `pip` to install PyAnsys Geometry.

```
pip install ansys-geometry-core
```


AVAILABLE MODES

This client library works with a Geometry service backend. There are several ways of running this backend, although the preferred and high-performance mode is using Docker containers. Select the option that suits your needs best.

Docker containers Launch the Geometry service as a Docker container and connect to it from PyAnsys Geometry.

Docker containers **Local service** Launch the Geometry service locally on your machine and connect to it from PyAnsys Geometry.

Launch a local session **Remote service** Launch the Geometry service on a remote machine and connect to it using PIM (Product Instance Manager).

Launch a remote session **Connect to an existing service** Connect to an existing Geometry service locally or remotely.

Use an existing session

COMPATIBILITY WITH ANSYS RELEASES

PyAnsys Geometry continues to evolve as the Ansys products move forward. For more information, see *Ansys product version compatibility*.

DEVELOPMENT INSTALLATION

In case you want to support the development of PyAnsys Geometry, install the repository in development mode. For more information, see *Install package in development mode*.

FREQUENTLY ASKED QUESTIONS

Any questions? Refer to *Q&A* before submitting an issue.

5.1 Docker containers

5.1.1 What is Docker?

Docker is an open platform for developing, shipping, and running apps in a containerized way.

Containers are standard units of software that package the code and all its dependencies so that the app runs quickly and reliably from one computing environment to another.

Ensure that the machine where the Geometry service is to run has Docker installed. Otherwise, see [Install Docker Engine](#) in the Docker documentation.

5.1.2 Select your Docker container

Currently, the Geometry service backend is mainly delivered as a **Windows** Docker container. However, these containers require a Windows machine to run them.

Select the kind of Docker container you want to build:

Windows Docker container Build a Windows Docker container for the Geometry service and use it from PyAnsys Geometry. Explore the full potential of the Geometry service.

Windows Docker container *Go to Getting started*

Windows Docker container

Contents

- *Windows Docker container*
 - *Docker for Windows containers*
 - *Build or install the Geometry service image*
 - * *GitHub Container Registry*
 - * *Build the Geometry service Windows container*
 - *Prerequisites*
 - *Build from available Ansys installation*

- *Build the Docker image from available binaries*
- *Launch the Geometry service*
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 - * *Geometry service launcher*
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Docker for Windows containers

To run the Windows Docker container for the Geometry service, ensure that you follow these steps when installing Docker:

1. Install [Docker Desktop](#).
2. When prompted for **Use WSL2 instead of Hyper-V (recommended)**, **clear** this checkbox. Hyper-V must be enabled to run Windows Docker containers.
3. Once the installation finishes, restart your machine and start Docker Desktop.
4. On the Windows taskbar, go to the **Show hidden icons** section, right-click in the Docker Desktop app, and select **Switch to Windows containers**.

Now that your Docker engine supports running Windows Docker containers, you can build or install the PyAnsys Geometry image.

Build or install the Geometry service image

There are two options for installing the PyAnsys Geometry image:

- Download it from the *GitHub Container Registry*.
- *Build the Geometry service Windows container*.

GitHub Container Registry

Note

This option is only available for users with write access to the repository or who are members of the Ansys organization.

Once Docker is installed on your machine, follow these steps to download the Windows Docker container for the Geometry service and install this image.

1. Using your GitHub credentials, download the Docker image from the [PyAnsys Geometry repository](#) on GitHub.
2. Use a GitHub personal access token with permission for reading packages to authorize Docker to access this repository. For more information, see [Managing your personal access tokens](#) in the GitHub documentation.
3. Save the token to a file with this command:

```
echo XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX > GH_TOKEN.txt
```

4. Authorize Docker to access the repository and run the commands for your OS. To see these commands, click the tab for your OS.

Powershell

```
$env:GH_USERNAME=<my-github-username>
cat GH_TOKEN.txt | docker login ghcr.io -u $env:GH_USERNAME --password-stdin
```

Windows CMD

```
SET GH_USERNAME=<my-github-username>
type GH_TOKEN.txt | docker login ghcr.io -u %GH_USERNAME% --password-stdin
```

5. Pull the Geometry service locally using Docker with a command like this:

```
docker pull ghcr.io/ansys/geometry:windows-latest
```

Build the Geometry service Windows container

The Geometry service Docker containers can be easily built by following these steps.

Inside the repository's docker folder, there are two Dockerfile files:

- `linux/Dockerfile`: Builds the Linux-based Docker image.
- `windows/Dockerfile`: Builds the Windows-based Docker image.

Depending on the characteristics of the Docker engine installed on your machine, either one or the other has to be built.

This guide focuses on building the `windows/Dockerfile` image.

There are two build modes:

- **Build from available Ansys installation**: This mode builds the Docker image using the Ansys installation available in the machine where the Docker image is being built.
- **Build from available binaries**: This mode builds the Docker image using the binaries available in the [ansys/pyansys-geometry-binaries](#) repository. If you do not have access to this repository, you can only use the first mode.

Prerequisites

- Ensure that Docker is installed in your machine. If you do not have Docker available, see *Docker for Windows containers*.

Build from available Ansys installation

To build your own image based on your own Ansys installation, follow these instructions:

- Download the [Python Docker build script](#).
- Execute the script with the following command (no specific location needed):

```
python build_docker_windows.py
```

Check that the image has been created successfully. You should see output similar to this:

```
docker images
>>> REPOSITORY                                TAG
↔   IMAGE ID      CREATED        SIZE
(continues on next page)
```


(continued from previous page)

```
>>> ghcr.io/ansys/geometry                                windows-*****
↵      ..... X seconds ago Y.ZZGB
>>> .....
↵      ..... .....
```

Build the Docker image from available binaries

Prior to building your image, follow these steps:

- Download the latest [Windows Dockerfile](#).
- Download the latest release artifacts for the [Windows Docker container \(ZIP file\)](#) for your version.

Note

Only users with access to <https://github.com/ansys/pyansys-geometry-binaries> can download these binaries.

- Move this ZIP file to the location of the Windows Dockerfile previously downloaded.

To build your image, follow these instructions:

1. Navigate to the folder where the ZIP file and Dockerfile are located.
2. Run this Docker command:

```
docker build -t ghcr.io/ansys/geometry:windows-latest -f windows/Dockerfile .
```

3. Check that the image has been created successfully. You should see output similar to this:

```
docker images
>>> REPOSITORY                                TAG
↵      IMAGE ID          CREATED          SIZE
>>> ghcr.io/ansys/geometry                    windows-*****
↵      ..... X seconds ago Y.ZZGB
>>> .....
↵      ..... .....
```

Launch the Geometry service

There are methods for launching the Geometry service:

- You can use the [PyAnsys Geometry launcher](#).
- You can manually launch the [Geometry service](#).

Environment variables

The Geometry service requires this mandatory environment variable for its use:

- **LICENSE_SERVER**: License server (IP address or DNS) that the Geometry service is to connect to. For example, 127.0.0.1.

You can also specify other optional environment variables:

- **ENABLE_TRACE**: Whether to set up the trace level for debugging purposes. The default is 0, in which case the trace level is not set up. Options are 1 and 0.

- **LOG_LEVEL**: Sets the Geometry service logging level. The default is 2, in which case the logging level is INFO.

Here are some terms to keep in mind:

- **host**: Machine that hosts the Geometry service. It is typically on localhost, but if you are deploying the service on a remote machine, you must pass in this host machine's IP address when connecting. By default, PyAnsys Geometry assumes it is on localhost.
- **port**: Port that exposes the Geometry service on the host machine. Its value is assumed to be 50051, but users can deploy the service on preferred ports.

Prior to using the PyAnsys Geometry launcher to launch the Geometry service, you must define general environment variables required for your OS. You do not need to define these environment variables prior to manually launching the Geometry service.

Using PyAnsys Geometry launcher

Define the following general environment variables prior to using the PyAnsys Geometry launcher. Click the tab for your OS to see the appropriate commands.

Linux/Mac

```
export ANSRV_GEO_LICENSE_SERVER=127.0.0.1
export ANSRV_GEO_ENABLE_TRACE=0
export ANSRV_GEO_LOG_LEVEL=2
export ANSRV_GEO_HOST=127.0.0.1
export ANSRV_GEO_PORT=50051
```

Powershell

```
$env:ANSRV_GEO_LICENSE_SERVER="127.0.0.1"
$env:ANSRV_GEO_ENABLE_TRACE=0
$env:ANSRV_GEO_LOG_LEVEL=2
$env:ANSRV_GEO_HOST="127.0.0.1"
$env:ANSRV_GEO_PORT=50051
```

Windows CMD

```
SET ANSRV_GEO_LICENSE_SERVER=127.0.0.1
SET ANSRV_GEO_ENABLE_TRACE=0
SET ANSRV_GEO_LOG_LEVEL=2
SET ANSRV_GEO_HOST=127.0.0.1
SET ANSRV_GEO_PORT=50051
```

Warning

When running a Windows Docker container, certain high-value ports might be restricted from its use. This means that the port exposed by the container has to be set to lower values. You should change the value of ANSRV_GEO_PORT to use a port such as 700, instead of 50051.

Manual launch

You do not need to define general environment variables prior to manually launching the Geometry service. They are directly passed to the Docker container itself.

Geometry service launcher

As mentioned earlier, you can launch the Geometry service locally in two different ways. To see the commands for each method, click the following tabs.

Using PyAnsys Geometry launcher

This method directly launches the Geometry service and provides a `Modeler` object.

```
from ansys.geometry.core.connection import launch_modeler

modeler = launch_modeler()
```

The `launch_modeler()` method launches the Geometry service under the default conditions. For more configurability, use the `launch_docker_modeler()` method.

Manual launch

This method requires that you manually launch the Geometry service. Remember to pass in the different environment variables that are needed. Afterwards, see the next section to understand how to connect to this service instance from PyAnsys Geometry.

Linux/Mac

```
docker run \
  --name ans_geo \
  -e LICENSE_SERVER=<LICENSE_SERVER> \
  -p 50051:50051 \
  ghcr.io/ansys/geometry:<TAG>
```

Powershell

```
docker run `
  --name ans_geo `
  -e LICENSE_SERVER=<LICENSE_SERVER> `
  -p 50051:50051 `
  ghcr.io/ansys/geometry:<TAG>
```

Windows CMD

```
docker run ^
  --name ans_geo ^
  -e LICENSE_SERVER=<LICENSE_SERVER> ^
  -p 50051:50051 ^
  ghcr.io/ansys/geometry:<TAG>
```

Warning

When running a Windows Docker container, certain high-value ports might be restricted from its use. This means that the port exposed by the container has to be set to lower values. You should change the value of `-p 50051:50051` to use a port such as `-p 700:50051`.

Connect to the Geometry service

After the Geometry service is launched, connect to it with these commands:

```
from ansys.geometry.core import Modeler

modeler = Modeler()
```

By default, the `Modeler` instance connects to `127.0.0.1` ("localhost") on port `50051`. You can change this by modifying the `host` and `port` parameters of the `Modeler` object, but note that you must also modify your docker run command by changing the `<HOST-PORT>-50051` argument.

The following tabs show the commands that set the environment variables and `Modeler` function.

Warning

When running a Windows Docker container, certain high-value ports might be restricted from its use. This means that the port exposed by the container has to be set to lower values. You should change the value of `ANSRV_GEO_PORT` to use a port such as `700`, instead of `50051`.

Environment variables**Linux/Mac**

```
export ANSRV_GEO_HOST=127.0.0.1
export ANSRV_GEO_PORT=50051
```

Powershell

```
$env:ANSRV_GEO_HOST="127.0.0.1"
$env:ANSRV_GEO_PORT=50051
```

Windows CMD

```
SET ANSRV_GEO_HOST=127.0.0.1
SET ANSRV_GEO_PORT=50051
```

Modeler function

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler(host="127.0.0.1", port=50051)
```

Go to Docker containers

Go to Getting started

5.2 Launch a local session

If Ansys 2024 R1 or later and PyAnsys Geometry are installed, you can create a local backend session using Discovery, SpaceClaim, or the Geometry service. Once the backend is running, PyAnsys Geometry can manage the connection.

To launch and establish a connection to the service, open Python and use the following commands for either Discovery, SpaceClaim, or the Geometry service.

Discovery

```
from ansys.geometry.core import launch_modeler_with_discovery

modeler = launch_modeler_with_discovery()
```

SpaceClaim

```
from ansys.geometry.core import launch_modeler_with_spaceclaim

modeler = launch_modeler_with_spaceclaim()
```

Geometry service

```
from ansys.geometry.core import launch_modeler_with_geometry_service

modeler = launch_modeler_with_geometry_service()
```

When launching via Geometry Service, if you have a custom local install, you can define the path of this install in the `ANSYS_GEOMETRY_SERVICE_ROOT` environment variable. In that case, the launcher uses this location by default.

Powershell

```
$env:ANSYS_GEOMETRY_SERVICE_ROOT="C:\Program Files\ANSYS Inc\v252\GeometryService"
# or
$env:ANSYS_GEOMETRY_SERVICE_ROOT="C:\myCustomPath\CoreGeometryService"
```

Windows CMD

```
SET ANSYS_GEOMETRY_SERVICE_ROOT="C:\Program Files\ANSYS Inc\v252\GeometryService"
# or
SET ANSYS_GEOMETRY_SERVICE_ROOT="C:\myCustomPath\CoreGeometryService"
```

Linux

```
export ANSYS_GEOMETRY_SERVICE_ROOT=/my_path/to/core_geometry_service
```

For more information on the arguments accepted by the launcher methods, see their API documentation:

- `launch_modeler_with_discovery`
- `launch_modeler_with_spaceclaim`
- `launch_modeler_with_geometry_service`

Note

Because this is the first release of the Geometry service, you cannot yet define a product version or API version.

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5.3 Launch a remote session

If a remote server is running Ansys 2024 R1 or later and is also running PIM (Product Instance Manager), you can use PIM to start a Discovery or SpaceClaim session that PyAnsys Geometry can connect to.

Warning

This option is only available for Ansys employees.

Only Ansys employees with credentials to the Artifact Repository Browser can download ZIP files for PIM.

5.3.1 Set up the client machine

1. To establish a connection to the existing session from your client machine, open Python and run these commands:

```
from ansys.discovery.core import launch_modeler_with_pimlight_and_discovery

disco = launch_modeler_with_pimlight_and_discovery("241")
```

The preceding commands launch a Discovery (version 24.1) session with the API server. You receive a model object back from Discovery that you then use as a PyAnsys Geometry client.

2. Start SpaceClaim or the Geometry service remotely using commands like these:

```
from ansys.discovery.core import launch_modeler_with_pimlight_and_spaceclaim

sc = launch_modeler_with_pimlight_and_spaceclaim("version")

from ansys.discovery.core import launch_modeler_with_pimlight_and_geometry_service

geo = launch_modeler_with_pimlight_and_geometry_service("version")
```

Note

Performing all these operations remotely eliminates the need to worry about the starting endpoint or managing the session.

5.3.2 End the session

To end the session, run the corresponding command:

```
disco.close()
sc.close()
geo.close()
```

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5.4 Use an existing session

If a session of Discovery, SpaceClaim, or the Geometry service is already running, PyAnsys Geometry can be used to connect to it.

Warning

Running a SpaceClaim or Discovery normal session does not suffice to be able to use it with PyAnsys Geometry. Both products need the ApiServer extension to be running. In this case, to ease the process, you should launch the products directly from the PyAnsys Geometry library as shown in *Launch a local session*.

5.4.1 Establish the connection

From Python, establish a connection to the existing client session by creating a `Modeler` object:

```
from ansys.geometry.core import Modeler

modeler = Modeler(host="localhost", port=50051)
```

If no error messages are received, your connection is established successfully. Note that your local port number might differ from the one shown in the preceding code.

5.4.2 Verify the connection

If you want to verify that the connection is successful, request the status of the client connection inside your `Modeler` object:

```
>>> modeler.client
Ansys Geometry Modeler Client (...)
Target:    localhost:50051
Connection: Healthy
```

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5.5 Ansys version compatibility

The following table summarizes the compatibility matrix between the PyAnsys Geometry service and the Ansys product versions.

PyAnsys Geometry versions	Ansys Product versions	Geometry Service (dockerized)	Geometry Service (standalone)	Discovery	SpaceClaim
0.2.X	23R2				
0.3.X	23R2 (partially)				
0.4.X	24R1 onward				
0.5.X	24R1 onward				

 **Tip**

Forth- and back-compatibility mechanism

Starting on version 0.5.X and onward, PyAnsys Geometry has implemented a forth- and back-compatibility mechanism to ensure that the Python library can be used with different versions of the Ansys products.

Methods are now decorated with the `@min_backend_version` decorator to indicate the compatibility with the Ansys product versions. If an unsupported method is called, a `GeometryRuntimeError` is raised when attempting to use the method. Users are informed of the minimum Ansys product version required to use the method.

Access to the documentation for the preceding versions is found at the [Versions](#) page.

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5.6 Install package in development mode

This topic assumes that you want to install PyAnsys Geometry in developer mode so that you can modify the source and enhance it. You can install PyAnsys Geometry from PyPI, Conda, or from the [PyAnsys Geometry repository](#) on GitHub.

Contents

- *Install package in development mode*
 - *Package dependencies*
 - *PyPI*
 - *Conda*
 - *GitHub*
 - *Install in offline mode*
 - *Verify your installation*

5.6.1 Package dependencies

PyAnsys Geometry is supported on Python version 3.10 and later. As indicated in the [Moving to require Python 3](#) statement, previous versions of Python are no longer supported.

PyAnsys Geometry dependencies are automatically checked when packages are installed. These projects are required dependencies for PyAnsys Geometry:

- [ansys-api-geometry](#): Used for supplying gRPC code generated from Protobuf (PROTO) files
- [NumPy](#): Used for data array access
- [Pint](#): Used for measurement units
- [PyVista](#): Used for interactive 3D plotting
- [SciPy](#): Used for geometric transformations

5.6.2 PyPI

Before installing PyAnsys Geometry, to ensure that you have the latest version of `pip`, run this command:

```
python -m pip install -U pip
```

Then, to install PyAnsys Geometry, run this command:

```
python -m pip install ansys-geometry-core
```

5.6.3 Conda

You can also install PyAnsys Geometry using `conda`. First, ensure that you have the latest version:

```
conda update -n base -c defaults conda
```

Then, to install PyAnsys Geometry, run this command:

```
conda install -c conda-forge ansys-geometry-core
```

5.6.4 GitHub

To install the latest release from the [PyAnsys Geometry repository](https://github.com/ansys/pyansys-geometry) on GitHub, run these commands:

```
git clone https://github.com/ansys/pyansys-geometry
cd pyansys-geometry
pip install -e .
```

To verify your development installation, run this command:

```
tox
```

5.6.5 Install in offline mode

If you lack an internet connection on your installation machine (or you do not have access to the private Ansys PyPI packages repository), you should install PyAnsys Geometry by downloading the wheelhouse archive for your corresponding machine architecture from the repository's [Releases page](#).

Each wheelhouse archive contains all the Python wheels necessary to install PyAnsys Geometry from scratch on Windows, Linux, and MacOS from Python 3.10 to 3.13. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

For example, on Linux with Python 3.10, unzip the wheelhouse archive and install it with these commands:

```
unzip ansys-geometry-core-v0.9.0-wheelhouse-ubuntu-3.10.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

If you are on Windows with Python 3.10, unzip the wheelhouse archive to a wheelhouse directory and then install using the same `pip install` command as in the preceding example.

Consider installing using a virtual environment. For more information, see [Creation of virtual environments](#) in the Python documentation.

5.6.6 Verify your installation

Verify the `Modeler()` connection with this code:

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> print(modeler)

Ansys Geometry Modeler (0x205c5c17d90)

Ansys Geometry Modeler Client (0x205c5c16e00)
Target:      localhost:652
Connection: Healthy
```

If you see a response from the server, you can start using PyAnsys Geometry as a service. For more information on PyAnsys Geometry usage, see *User guide*.

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5.7 Frequently asked questions

5.7.1 What is PyAnsys?

PyAnsys is a set of open source Python libraries that allow you to interface with Ansys Electronics Desktop (AEDT), Ansys Mechanical, Ansys Parametric Design Language (APDL), Ansys Fluent, and other Ansys products.

You can use PyAnsys libraries within a Python environment of your choice in conjunction with external Python libraries.

5.7.2 How is the Ansys Geometry Service installed?

Note

This question is answered in <https://github.com/ansys/pyansys-geometry/issues/1022> and <https://github.com/ansys/pyansys-geometry/discussions/883>

The Ansys Geometry service is available as a standalone service and it is installed through the Ansys unified installer or the automated installer. Both are available for download from the [Ansys Customer Portal](#).

When using the unified or automated installer, it is necessary to pass in the `-geometryservice` flag to install it.

Overall, the command to install the Ansys Geometry service with the unified installer is:

```
setup.exe -silent -geometryservice
```

You can verify that the installation was successful by checking whether the product has been installed on your file directory. If you are using the default installation directory, the product is installed in the following directory:

```
C:\Program Files\ANSYS Inc\vXXX\GeometryService
```

Where `vXXX` is the Ansys version that you have installed.

5.7.3 What Ansys license is needed to run the Geometry service?

Note

This question is answered in <https://github.com/ansys/pyansys-geometry/discussions/754>.

The Ansys Geometry service is a headless service developed on top of the modeling libraries for Discovery and SpaceClaim.

Both in its standalone and Docker versions, the Ansys Geometry service requires a **Discovery Modeling** license to run.

To run PyAnsys Geometry against other backends, such as Discovery or SpaceClaim, users must have an Ansys license that allows them to run these Ansys products.

The **Discovery Modeling** license is one of these licenses, but there are others, such as the Ansys Mechanical Enterprise license, that also allow users to run these Ansys products. However, the Geometry service is only compatible with the **Discovery Modeling** license.

5.7.4 How to build the Docker image for the Ansys Geometry service?

Note

This question is answered in <https://github.com/ansys/pyansys-geometry/discussions/883>

To build your own Docker image for the Ansys Geometry service, users should follow the instructions provided in *Build from available Ansys installation*. The resulting image is a Windows-based Docker image that contains the Ansys Geometry service.

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This section provides an overview of the PyAnsys Geometry library, explaining key concepts and approaches for primitives, sketches (2D basic shape elements), and model designs.

6.1 Primitives

The PyAnsys Geometry *math* subpackage consists of primitive representations of basic geometric objects, such as a point, vector, and matrix. To operate and manipulate physical quantities, this subpackage uses [Pint](#), a third-party open source software that other PyAnsys libraries also use. It also uses its *shapes* subpackage to evaluate and represent geometric shapes (both curves and surfaces), such as lines, circles, cones, spheres and torus.

This table shows PyAnsys Geometry names and base values for the physical quantities:

Name	value
LENGTH_ACCURACY	1e-8
ANGLE_ACCURACY	1e-6
DEFAULT_UNITS.LENGTH	meter
DEFAULT_UNITS.ANGLE	radian

To define accuracy and measurements, you use these PyAnsys Geometry classes:

- *Accuracy()*
- *Measurements()*

6.1.1 Planes

The *Plane()* class provides primitive representation of a 2D plane in 3D space. It has an origin and a coordinate system. Sketched shapes are always defined relative to a plane. The default working plane is XY, which has (0, 0) as its origin.

If you create a 2D object in the plane, PyAnsys Geometry converts it to the global coordinate system so that the 2D feature executes as expected:

```
from ansys.geometry.core.math import Plane, Point3D, UnitVector3D

origin = Point3D([42, 99, 13])
plane = Plane(origin, UnitVector3D([1, 0, 0]), UnitVector3D([0, 1, 0]))
```

6.2 Sketch

The PyAnsys Geometry *sketch* subpackage is used to build 2D basic shapes. Shapes consist of two fundamental constructs:

- **Edge:** A connection between two or more 2D points along a particular path. An edge represents an open shape such as an arc or line.
- **Face:** A set of edges that enclose a surface. A face represents a closed shape such as a circle or triangle.

To initialize a sketch, you first specify the *Plane()* class, which represents the plane in space from which other PyAnsys Geometry objects can be located.

This code shows how to initialize a sketch:

```
from ansys.geometry.core.sketch import Sketch

sketch = Sketch()
```

You then construct a sketch, which can be done using different approaches.

6.2.1 Functional-style API

A functional-style API is sometimes called a *fluent functional-style api* or *fluent API* in the developer community. However, to avoid confusion with the Ansys Fluent product, the PyAnsys Geometry documentation refrains from using the latter terms.

One of the key features of a functional-style API is that it keeps an active context based on the previously created edges to use as a reference starting point for additional objects.

The following code creates a sketch with its origin as a starting point. Subsequent calls create segments, which take as a starting point the last point of the previous edge.

```
from ansys.geometry.core.math import Point2D

sketch.segment_to_point(Point2D([3, 3]), "Segment2").segment_to_point(
    Point2D([3, 2]), "Segment3"
)
sketch.plot()
```

A functional-style API is also able to get a desired shape of the sketch object by taking advantage of user-defined labels:

```
sketch.get("Segment2")
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

6.2.2 Direct API

A direct API is sometimes called an *element-based approach* in the developer community.

This code shows how you can use a direct API to create multiple elements independently and combine them all together in a single plane:

```
sketch.triangle(
    Point2D([-10, 10]), Point2D([5, 6]), Point2D([-10, -10]), tag="triangle2"
```

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```
)
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
↳http-equiv="Content-...
```

For more information on sketch shapes, see the `Sketch()` subpackage.

6.3 Designer

The PyAnsys Geometry *designer* subpackage organizes geometry assemblies and synchronizes to a supporting Geometry service instance.

6.3.1 Create the model

This code create the `Modeler()` object which owns the whole designs tools and data.

```
from ansys.geometry.core import Modeler

# Create the modeler object itself
modeler = Modeler()
```

6.3.2 Define the model

The following code define the model by creating a sketch with a circle on the client. It then creates the model on the server.

```
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from pint import Quantity

# Create a sketch and draw a circle on the client
sketch = Sketch()
sketch.circle(Point2D([10, 10], UNITS.mm), Quantity(10, UNITS.mm))

# Create your design on the server
design_name = "ExtrudeProfile"
design = modeler.create_design(design_name)
```

6.3.3 Add materials to model

This code adds the data structure and properties for individual materials:

```
from ansys.geometry.core.materials.material import Material
from ansys.geometry.core.materials.property import (
    MaterialProperty,
    MaterialPropertyType,
)

density = Quantity(125, 1000 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
```

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```
poisson_ratio = Quantity(0.33, UNITS.dimensionless)
tensile_strength = Quantity(45)
material = Material(
    "steel",
    density,
    [MaterialProperty(MaterialPropertyType.POISSON_RATIO, "myPoisson", poisson_ratio)],
)
material.add_property(MaterialPropertyType.TENSILE_STRENGTH, "myTensile", Quantity(45))
design.add_material(material)
```

6.3.4 Create bodies by extruding the sketch

Extruding a sketch projects all of the specified geometries onto the body. To create a solid body, this code extrudes the sketch profile by a given distance.

```
body = design.extrude_sketch("JustACircle", sketch, Quantity(10, UNITS.mm))
```

6.3.5 Create bodies by extruding the face

The following code shows how you can also extrude a face profile by a given distance to create a solid body. There are no modifications against the body containing the source face.

```
longer_body = design.extrude_face(
    "LongerCircleFace", body.faces[0], Quantity(20, UNITS.mm)
)
```

You can also translate and tessellate design bodies and project curves onto them. For more information, see these classes:

- `Body()`
- `Component()`

6.3.6 Download and save design

You can save your design to disk or download the design of the active Geometry server instance. The following code shows how to download and save the design.

```
file = "path/to/download.scdocx"
design.download(file)
```

For more information, see the *Design* submodule.

6.4 PyAnsys Geometry overview

PyAnsys Geometry is a Python wrapper for the Ansys Geometry service. Here are some of the key features of PyAnsys Geometry:

- Ability to use the library alongside other Python libraries
- A *functional-style* API for a clean and easy coding experience
- Built-in examples

6.5 Simple interactive example

This simple interactive example shows how to start an instance of the Geometry server and create a geometry model.

6.5.1 Start Geometry server instance

The `Modeler()` class within the `ansys-geometry-core` library creates an instance of the Geometry service. By default, the `Modeler` instance connects to `127.0.0.1` ("localhost") on port `50051`. You can change this by modifying the `host` and `port` parameters of the `Modeler` object, but note that you must also modify your `docker run` command by changing the `<HOST-PORT>:50051` argument.

This code starts an instance of the Geometry service:

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
```

6.5.2 Create geometry model

Once an instance has started, you can create a geometry model by initializing the `sketch` subpackage and using the `shapes` subpackage.

```
from ansys.geometry.core.math import Plane, Point3D, Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch

# Define our sketch
origin = Point3D([0, 0, 10])
plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 1, 0])

# Create the sketch
sketch = Sketch(plane)
sketch.circle(Point2D([1, 1]), 30 * UNITS.m)
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```


API REFERENCE

This section describes ansys-geometry-core endpoints, their capabilities, and how to interact with them programmatically.

7.1 The `ansys.geometry.core` library

7.1.1 Summary

Subpackages

<i>connection</i>	PyAnsys Geometry connection subpackage.
<i>designer</i>	PyAnsys Geometry designer subpackage.
<i>materials</i>	PyAnsys Geometry materials subpackage.
<i>math</i>	PyAnsys Geometry math subpackage.
<i>misc</i>	Provides the PyAnsys Geometry miscellaneous subpackage.
<i>parameters</i>	PyAnsys Geometry parameters subpackage.
<i>plotting</i>	Provides the PyAnsys Geometry plotting subpackage.
<i>shapes</i>	Provides the PyAnsys Geometry <code>geometry</code> subpackage.
<i>sketch</i>	PyAnsys Geometry sketch subpackage.
<i>tools</i>	PyAnsys Geometry tools subpackage.

Submodules

<i>errors</i>	Provides PyAnsys Geometry-specific errors.
<i>logger</i>	Provides a general framework for logging in PyAnsys Geometry.
<i>modeler</i>	Provides for interacting with the Geometry service.
<i>typing</i>	Provides typing of values for PyAnsys Geometry.

Attributes

<code>__version__</code>	PyAnsys Geometry version.
--------------------------	---------------------------

Constants

<code>USE_SERVICE_COLORS</code>	Global constant for checking whether to use service colors for plotting
<code>DISABLE_MULTIPLE_DESIGN_CHECK</code>	Deprecated constant. Use <code>DISABLE_ACTIVE_DESIGN_CHECK</code> instead.
<code>DISABLE_ACTIVE_DESIGN_CHECK</code>	Global constant for disabling the <code>ensure_design_is_active</code> check.
<code>DOCUMENTATION_BUILD</code>	Global flag for the documentation to use the proper PyVista Jupyter backend.

The connection package

Summary

Submodules

<code>backend</code>	Module providing definitions for the backend types.
<code>client</code>	Module providing a wrapped abstraction of the gRPC stubs.
<code>conversions</code>	Module providing for conversions.
<code>defaults</code>	Module providing default connection parameters.
<code>docker_instance</code>	Module for connecting to a local Geometry Service Docker container.
<code>launcher</code>	Module for connecting to instances of the Geometry service.
<code>product_instance</code>	Module containing the <code>ProductInstance</code> class.
<code>validate</code>	Module to perform a connection validation check.

The backend.py module

Summary

Enums

<code>BackendType</code>	Provides an enum holding the available backend types.
<code>ApiVersions</code>	Provides an enum for all the compatibles API versions.

BackendType

`class ansys.geometry.core.connection.backend.BackendType(*args, **kws)`

Bases: `enum.Enum`

Provides an enum holding the available backend types.

Overview

Attributes

<code>DISCOVERY</code>
<code>SPACECLAIM</code>
<code>WINDOWS_SERVICE</code>
<code>LINUX_SERVICE</code>
<code>CORE_WINDOWS</code>
<code>CORE_LINUX</code>
<code>DISCOVERY_HEADLESS</code>

Static methods

<code>is_core_service</code>	Determine whether the backend is CoreService based or not.
------------------------------	--

Import detail

```
from ansys.geometry.core.connection.backend import BackendType
```

Attribute detail

BackendType.DISCOVERY = 0

BackendType.SPACECLAIM = 1

BackendType.WINDOWS_SERVICE = 2

BackendType.LINUX_SERVICE = 3

BackendType.CORE_WINDOWS = 4

BackendType.CORE_LINUX = 5

BackendType.DISCOVERY_HEADLESS = 6

Method detail

static BackendType.is_core_service(*backend_type*: BackendType) → bool

Determine whether the backend is CoreService based or not.

Parameters

backend_type

[BackendType] The backend type to check whether or not it's a CoreService type.

Returns

bool

True if the backend is CoreService based, False otherwise.

ApiVersions

class ansys.geometry.core.connection.backend.ApiVersions(*args, **kws)

Bases: `enum.Enum`

Provides an enum for all the compatibles API versions.

Overview

Attributes

V_21
V_22
V_231
V_232
V_241
V_242
V_251

Static methods

<code>parse_input</code>	Convert an input to an ApiVersions enum.
--------------------------	--

Import detail

```
from ansys.geometry.core.connection.backend import ApiVersions
```

Attribute detail

`ApiVersions.V_21 = 21`

`ApiVersions.V_22 = 22`

`ApiVersions.V_231 = 231`

`ApiVersions.V_232 = 232`

`ApiVersions.V_241 = 241`

`ApiVersions.V_242 = 242`

`ApiVersions.V_251 = 251`

Method detail

static `ApiVersions.parse_input(version: int | str | ApiVersions) → ApiVersions`

Convert an input to an ApiVersions enum.

Parameters

version

`[int | str | ApiVersions]` The version to convert to an ApiVersions enum.

Returns

ApiVersions

The version as an ApiVersions enum.

Description

Module providing definitions for the backend types.

The client.py module

Summary

Classes

<i>GrpcClient</i>	Wraps the gRPC connection for the Geometry service.
-------------------	---

Functions

<i>wait_until_healthy</i>	Wait until a channel is healthy before returning.
---------------------------	---

GrpcClient

```
class ansys.geometry.core.connection.client.GrpcClient(host: str = DEFAULT_HOST, port: str | int
= DEFAULT_PORT, channel: grpc.Channel
| None = None, remote_instance: an-
sys.platform.instancemanagement.Instance |
None = None, docker_instance: an-
sys.geometry.core.connection.docker_instance.LocalDockerInst
| None = None, product_instance: an-
sys.geometry.core.connection.product_instance.ProductInstanc
| None = None, timeout:
ansys.geometry.core.typing.Real = 120,
logging_level: int = logging.INFO,
logging_file: pathlib.Path | str | None =
None, backend_type: an-
sys.geometry.core.connection.backend.BackendType
| None = None)
```

Wraps the gRPC connection for the Geometry service.

Parameters

host

[str, default: DEFAULT_HOST] Host where the server is running.

port

[str or int, default: DEFAULT_PORT] Port number where the server is running.

channel

[Channel, default: None] gRPC channel for server communication.

remote_instance

[ansys.platform.instancemanagement.Instance, default: None] Corresponding re-
mote instance when the Geometry service is launched through PyPIM. This instance is
deleted when calling the GrpcClient.close method.

docker_instance

[LocalDockerInstance, default: None] Corresponding local Docker instance when the
Geometry service is launched using the launch_docker_modeler() method. This local
Docker instance is deleted when the GrpcClient.close method is called.

product_instance

[ProductInstance, default: None] Corresponding local prod-
uct instance when the product (Discovery or SpaceClaim) is

launched through the `launch_modeler_with_geometry_service()`, `launch_modeler_with_discovery()` or the `launch_modeler_with_spaceclaim()` interface. This instance will be deleted when the `GrpcClient.close` method is called.

timeout

[real, default: 120] Maximum time to spend trying to make the connection.

logging_level

[int, default: INFO] Logging level to apply to the client.

logging_file

[str or Path, default: None] File to output the log to, if requested.

backend_type: BackendType, default: None

Type of the backend that PyAnsys Geometry is communicating with. By default, this value is unknown, which results in None being the default value.

Overview

Methods

<code>close</code>	Close the channel.
<code>target</code>	Get the target of the channel.
<code>get_name</code>	Get the target name of the connection.

Properties

<code>backend_type</code>	Backend type.
<code>backend_version</code>	Get the current backend version.
<code>multiple_designs_allowed</code>	Flag indicating whether multiple designs are allowed.
<code>channel</code>	Client gRPC channel.
<code>log</code>	Specific instance logger.
<code>is_closed</code>	Flag indicating whether the client connection is closed.
<code>healthy</code>	Flag indicating whether the client channel is healthy.

Special methods

<code>__repr__</code>	Represent the client as a string.
-----------------------	-----------------------------------

Import detail

```
from ansys.geometry.core.connection.client import GrpcClient
```

Property detail

property `GrpcClient.backend_type`: `ansys.geometry.core.connection.backend.BackendType`
 Backend type.
 Options are `Windows Service`, `Linux Service`, `Discovery`, and `SpaceClaim`.

Notes

This method might return None because determining the backend type is not straightforward.

property GrpcClient.**backend_version**: **semver.version.Version**

Get the current backend version.

Returns

Version

Backend version.

property GrpcClient.**multiple_designs_allowed**: **bool**

Flag indicating whether multiple designs are allowed.

Notes

Currently, only one design is allowed per service. This method will always return False.

property GrpcClient.**channel**: **grpc.Channel**

Client gRPC channel.

property GrpcClient.**log**: **ansys.geometry.core.logger.PyGeometryCustomAdapter**

Specific instance logger.

property GrpcClient.**is_closed**: **bool**

Flag indicating whether the client connection is closed.

property GrpcClient.**healthy**: **bool**

Flag indicating whether the client channel is healthy.

Method detail

GrpcClient.**__repr__**() → **str**

Represent the client as a string.

GrpcClient.**close**()

Close the channel.

Notes

If an instance of the Geometry service was started using **PyPIM**, this instance is deleted. Furthermore, if a local Docker instance of the Geometry service was started, it is stopped.

GrpcClient.**target**() → **str**

Get the target of the channel.

GrpcClient.**get_name**() → **str**

Get the target name of the connection.

Description

Module providing a wrapped abstraction of the gRPC stubs.

Module detail

`client.wait_until_healthy(channel: grpc.Channel, timeout: float)`

Wait until a channel is healthy before returning.

Parameters

channel

[`Channel`] Channel that must be established and healthy.

timeout

[`float`] Timeout in seconds. Attempts are made with the following backoff strategy:

- Starts with 0.1 seconds.
- If the attempt fails, double the timeout.
- This is repeated until the next timeout exceeds the value for the remaining time. In that case, a final attempt is made with the remaining time.
- If the total elapsed time exceeds the value for the `timeout` parameter, a `TimeoutError` is raised.

Raises

TimeoutError

Raised when the total elapsed time exceeds the value for the `timeout` parameter.

The `conversions.py` module

Summary

Functions

<code>unit_vector_to_grpc_direction</code>	Convert a <code>UnitVector3D</code> class to a unit vector gRPC message.
<code>frame_to_grpc_frame</code>	Convert a <code>Frame</code> class to a frame gRPC message.
<code>plane_to_grpc_plane</code>	Convert a <code>Plane</code> class to a plane gRPC message.
<code>sketch_shapes_to_grpc_geometries</code>	Convert lists of <code>SketchEdge</code> and <code>SketchFace</code> to a gRPC message.
<code>sketch_edges_to_grpc_geometries</code>	Convert a list of <code>SketchEdge</code> to a gRPC message.
<code>sketch_arc_to_grpc_arc</code>	Convert an <code>Arc</code> class to an arc gRPC message.
<code>sketch_ellipse_to_grpc_ellipse</code>	Convert a <code>SketchEllipse</code> class to an ellipse gRPC message.
<code>sketch_circle_to_grpc_circle</code>	Convert a <code>SketchCircle</code> class to a circle gRPC message.
<code>point3d_to_grpc_point</code>	Convert a <code>Point3D</code> class to a point gRPC message.
<code>grpc_point_to_point3d</code>	Convert a point gRPC message class to a <code>Point3D</code> class.
<code>point2d_to_grpc_point</code>	Convert a <code>Point2D</code> class to a point gRPC message.
<code>sketch_polygon_to_grpc_polygon</code>	Convert a <code>Polygon</code> class to a polygon gRPC message.
<code>sketch_segment_to_grpc_line</code>	Convert a <code>Segment</code> class to a line gRPC message.
<code>tess_to_pd</code>	Convert an <code>ansys.api.geometry.Tessellation</code> to <code>pyvista.PolyData</code> .
<code>grpc_matrix_to_matrix</code>	Convert an <code>ansys.api.geometry.Matrix</code> to a <code>Matrix44</code> .
<code>grpc_frame_to_frame</code>	Convert a frame gRPC message to a <code>Frame</code> class.
<code>grpc_surface_to_surface</code>	Convert a surface gRPC message to a <code>Surface</code> class.
<code>grpc_curve_to_curve</code>	Convert a curve gRPC message to a <code>Curve</code> .
<code>curve_to_grpc_curve</code>	Convert a <code>Curve</code> object to a curve gRPC message.
<code>trimmed_curve_to_grpc_trimmed_curve</code>	Convert a <code>TrimmedCurve</code> to a trimmed curve gRPC message.
<code>surface_to_grpc_surface</code>	Convert a <code>Surface</code> object to a surface gRPC message.
<code>trimmed_surface_to_grpc_trimmed_surfa</code>	Convert a <code>TrimmedSurface</code> to a trimmed surface gRPC message.
<code>line_to_grpc_line</code>	Convert a <code>Line</code> to a line gRPC message.
<code>grpc_material_to_material</code>	Convert a material gRPC message to a <code>Material</code> class.
<code>grpc_material_property_to_material_pr</code>	Convert a material property gRPC message to a <code>MaterialProperty</code> class.

Description

Module providing for conversions.

Module detail

`conversions.unit_vector_to_grpc_direction`(*unit_vector*: `ansys.geometry.core.math.vector.UnitVector3D`)
→ `ansys.api.geometry.v0.models_pb2.Direction`

Convert a `UnitVector3D` class to a unit vector gRPC message.

Parameters

unit_vector
[`UnitVector3D`] Source vector data.

Returns

GRPCDirection
Geometry service gRPC direction message.

`conversions.frame_to_grpc_frame`(*frame*: `ansys.geometry.core.math.frame.Frame`) →
`ansys.api.geometry.v0.models_pb2.Frame`

Convert a `Frame` class to a frame gRPC message.

Parameters

frame

[Frame] Source frame data.

Returns

GRPCFrame

Geometry service gRPC frame message. The unit for the frame origin is meters.

`conversions.plane_to_grpc_plane(plane: ansys.geometry.core.math.plane.Plane) → ansys.api.geometry.v0.models_pb2.Plane`

Convert a Plane class to a plane gRPC message.

Parameters

plane

[Plane] Source plane data.

Returns

GRPCPlane

Geometry service gRPC plane message. The unit is meters.

`conversions.sketch_shapes_to_grpc_geometries(plane: ansys.geometry.core.math.plane.Plane, edges: list[ansys.geometry.core.sketch.edge.SketchEdge], faces: list[ansys.geometry.core.sketch.face.SketchFace], only_one_curve: bool = False) → ansys.api.geometry.v0.models_pb2.Geometries`

Convert lists of SketchEdge and SketchFace to a gRPC message.

Parameters

plane

[Plane] Plane for positioning the 2D sketches.

edges

[list[SketchEdge]] Source edge data.

faces

[list[SketchFace]] Source face data.

only_one_curve

[bool, default: False] Whether to project one curve of the whole set of geometries to enhance performance.

Returns

GRPCGeometries

Geometry service gRPC geometries message. The unit is meters.

`conversions.sketch_edges_to_grpc_geometries(edges: list[ansys.geometry.core.sketch.edge.SketchEdge], plane: ansys.geometry.core.math.plane.Plane) → tuple[list[ansys.api.geometry.v0.models_pb2.Line], list[ansys.api.geometry.v0.models_pb2.Arc]]`

Convert a list of SketchEdge to a gRPC message.

Parameters

edges

[list[SketchEdge]] Source edge data.

plane

[Plane] Plane for positioning the 2D sketches.

Returns

tuple[**list**[GRPCLine], **list**[GRPCArc]]

Geometry service gRPC line and arc messages. The unit is meters.

`conversions.sketch_arc_to_grpc_arc(arc: ansys.geometry.core.sketch.arc.Arc, plane: ansys.geometry.core.math.plane.Plane) → ansys.api.geometry.v0.models_pb2.Arc`

Convert an Arc class to an arc gRPC message.

Parameters

arc

[Arc] Source arc data.

plane

[Plane] Plane for positioning the arc within.

Returns

GRPCArc

Geometry service gRPC arc message. The unit is meters.

`conversions.sketch_ellipse_to_grpc_ellipse(ellipse: ansys.geometry.core.sketch.ellipse.SketchEllipse, plane: ansys.geometry.core.math.plane.Plane) → ansys.api.geometry.v0.models_pb2.Ellipse`

Convert a SketchEllipse class to an ellipse gRPC message.

Parameters

ellipse

[SketchEllipse] Source ellipse data.

Returns

GRPCEllipse

Geometry service gRPC ellipse message. The unit is meters.

`conversions.sketch_circle_to_grpc_circle(circle: ansys.geometry.core.sketch.circle.SketchCircle, plane: ansys.geometry.core.math.plane.Plane) → ansys.api.geometry.v0.models_pb2.Circle`

Convert a SketchCircle class to a circle gRPC message.

Parameters

circle

[SketchCircle] Source circle data.

plane

[Plane] Plane for positioning the circle.

Returns

GRPCCircle

Geometry service gRPC circle message. The unit is meters.

`conversions.point3d_to_grpc_point(point: ansys.geometry.core.math.point.Point3D) → ansys.api.geometry.v0.models_pb2.Point`

Convert a Point3D class to a point gRPC message.

Parameters

point

[Point3D] Source point data.

Returns

GRPCPoint

Geometry service gRPC point message. The unit is meters.

`conversions.grpc_point_to_point3d`(*point*: *ansys.api.geometry.v0.models_pb2.Point*) → *ansys.geometry.core.math.point.Point3D*

Convert a point gRPC message class to a Point3D class.

Parameters

point

[GRPCPoint] Source point data.

Returns

Point3D

Converted point.

`conversions.point2d_to_grpc_point`(*plane*: *ansys.geometry.core.math.plane.Plane*, *point2d*: *ansys.geometry.core.math.point.Point2D*) → *ansys.api.geometry.v0.models_pb2.Point*

Convert a Point2D class to a point gRPC message.

Parameters

plane

[Plane] Plane for positioning the 2D point.

point

[Point2D] Source point data.

Returns

GRPCPoint

Geometry service gRPC point message. The unit is meters.

`conversions.sketch_polygon_to_grpc_polygon`(*polygon*: *ansys.geometry.core.sketch.polygon.Polygon*, *plane*: *ansys.geometry.core.math.plane.Plane*) → *ansys.api.geometry.v0.models_pb2.Polygon*

Convert a Polygon class to a polygon gRPC message.

Parameters

polygon

[Polygon] Source polygon data.

Returns

GRPCPolygon

Geometry service gRPC polygon message. The unit is meters.

`conversions.sketch_segment_to_grpc_line`(*segment*: *ansys.geometry.core.sketch.segment.SketchSegment*, *plane*: *ansys.geometry.core.math.plane.Plane*) → *ansys.api.geometry.v0.models_pb2.Line*

Convert a Segment class to a line gRPC message.

Parameters

segment

[SketchSegment] Source segment data.

Returns

GRPCLine

Geometry service gRPC line message. The unit is meters.

`conversions.tess_to_pd(tess: ansys.api.geometry.v0.models_pb2.Tessellation) → pyvista.PolyData`

Convert an `ansys.api.geometry.Tessellation` to `pyvista.PolyData`.

`conversions.grpc_matrix_to_matrix(m: ansys.api.geometry.v0.models_pb2.Matrix) → ansys.geometry.core.math.matrix.Matrix44`

Convert an `ansys.api.geometry.Matrix` to a `Matrix44`.

`conversions.grpc_frame_to_frame(frame: ansys.api.geometry.v0.models_pb2.Frame) → ansys.geometry.core.math.frame.Frame`

Convert a frame gRPC message to a `Frame` class.

Parameters

GRPCFrame

Geometry service gRPC frame message. The unit for the frame origin is meters.

Returns

frame

[Frame] Resulting converted frame.

`conversions.grpc_surface_to_surface(surface: ansys.api.geometry.v0.models_pb2.Surface, surface_type: ansys.geometry.core.designer.face.SurfaceType) → ansys.geometry.core.shapes_surfaces.surface.Surface`

Convert a surface gRPC message to a `Surface` class.

Parameters

surface

[GRPCSurface] Geometry service gRPC surface message.

Returns

Surface

Resulting converted surface.

`conversions.grpc_curve_to_curve(curve: ansys.api.geometry.v0.models_pb2.CurveGeometry) → ansys.geometry.core.shapes_curves.curve.Curve`

Convert a curve gRPC message to a `Curve`.

Parameters

curve

[GRPCCurve] Geometry service gRPC curve message.

Returns

Curve

Resulting converted curve.

`conversions.curve_to_grpc_curve(curve: ansys.geometry.core.shapes_curves.curve.Curve) → ansys.api.geometry.v0.models_pb2.CurveGeometry`

Convert a `Curve` object to a curve gRPC message.

Parameters

curve

[Curve] Curve to convert.

Returns

GRPCCurve

Return Curve as a `ansys.api.geometry.CurveGeometry` message.

`conversions.trimmed_curve_to_grpc_trimmed_curve`(*curve*: `ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve`)
 → `ansys.api.geometry.v0.models_pb2.TrimmedCurve`

Convert a `TrimmedCurve` to a trimmed curve gRPC message.

Parameters

curve

[TrimmedCurve] Curve to convert.

Returns

GRPCTrimmedCurve

Geometry service gRPC `TrimmedCurve` message.

`conversions.surface_to_grpc_surface`(*surface*: `ansys.geometry.core.shapes-surfaces.surface.Surface`) →
`tuple[ansys.api.geometry.v0.models_pb2.Surface,`
`ansys.api.geometry.v0.models_pb2.SurfaceType]`

Convert a `Surface` object to a surface gRPC message.

Parameters

surface

[Surface] Surface to convert.

Returns

GRPCSurface

Return `Surface` as a `ansys.api.geometry.Surface` message.

GRPCSurfaceType

Return the gRPC surface type of `Surface`.

`conversions.trimmed_surface_to_grpc_trimmed_surface`(*trimmed_surface*: `ansys.geometry.core.shapes-surfaces.TrimmedSurface`)
 →
`ansys.api.geometry.v0.models_pb2.TrimmedSurface`

Convert a `TrimmedSurface` to a trimmed surface gRPC message.

Parameters

trimmed_surface

[TrimmedSurface] Surface to convert.

Returns

GRPCTrimmedSurface

Geometry service gRPC `TrimmedSurface` message.

`conversions.line_to_grpc_line`(*line*: `ansys.geometry.core.shapes-curves.line.Line`) →
`ansys.api.geometry.v0.models_pb2.Line`

Convert a `Line` to a line gRPC message.

Parameters

line
[Line] Line to convert.

Returns

GRPCLine
Geometry service gRPC Line message.

`conversions.grpc_material_to_material(material: ansys.api.geometry.v0.models_pb2.Material) → ansys.geometry.core.materials.material.Material`

Convert a material gRPC message to a `Material` class.

Parameters

material
[GRPCMaterial] Material gRPC message.

Returns

Material
Converted material.

`conversions.grpc_material_property_to_material_property(material_property: ansys.api.geometry.v0.models_pb2.MaterialProperty) → ansys.geometry.core.materials.material.MaterialProperty`

Convert a material property gRPC message to a `MaterialProperty` class.

Parameters

material_property
[GRPCMaterialProperty] Material property gRPC message.

Returns

MaterialProperty
Converted material property.

The defaults.py module

Summary

Constants

<code>DEFAULT_HOST</code>	Default for the HOST name.
<code>DEFAULT_PORT</code>	Default for the HOST port.
<code>MAX_MESSAGE_LENGTH</code>	Default for the gRPC maximum message length.
<code>GEOMETRY_SERVICE_DOCKER_IMAGE</code>	Default for the Geometry service Docker image location.
<code>DEFAULT_PIM_CONFIG</code>	Default for the PIM configuration when running PIM Light.

Description

Module providing default connection parameters.

Module detail

`defaults.DEFAULT_HOST`

Default for the HOST name.

By default, PyAnsys Geometry searches for the environment variable `ANSRV_GEO_HOST`, and if this variable does not exist, PyAnsys Geometry uses `127.0.0.1` as the host.

`defaults.DEFAULT_PORT: int`

Default for the HOST port.

By default, PyAnsys Geometry searches for the environment variable `ANSRV_GEO_PORT`, and if this variable does not exist, PyAnsys Geometry uses `50051` as the port.

`defaults.MAX_MESSAGE_LENGTH`

Default for the gRPC maximum message length.

By default, PyAnsys Geometry searches for the environment variable `PYGEOMETRY_MAX_MESSAGE_LENGTH`, and if this variable does not exist, it uses `256Mb` as the maximum message length.

`defaults.GEOMETRY_SERVICE_DOCKER_IMAGE = 'ghcr.io/ansys/geometry'`

Default for the Geometry service Docker image location.

Tag is dependent on what OS service is requested.

`defaults.DEFAULT_PIM_CONFIG`

Default for the PIM configuration when running PIM Light.

This parameter is only to be used when PIM Light is being run.

The `docker_instance.py` module

Summary

Classes

<i>LocalDockerInstance</i>	Instantiates a Geometry service as a local Docker container.
----------------------------	--

Enums

<i>GeometryContainers</i>	Provides an enum holding the available Geometry services.
---------------------------	---

Functions

<i>get_geometry_container_type</i>	Provide back the <i>GeometryContainers</i> value.
------------------------------------	---

`LocalDockerInstance`

```
class ansys.geometry.core.connection.docker_instance.LocalDockerInstance(port: int =
    DEFAULT_PORT,
    connect_to_existing_service:
    bool = True,
    restart_if_existing_service:
    bool = False, name:
    str | None = None,
    image:
    GeometryContainers
    | None = None)
```

Instantiates a Geometry service as a local Docker container.

By default, if a container with the Geometry service already exists at the given port, PyAnsys Geometry connects to it. Otherwise, PyAnsys Geometry tries to launch its own service.

Parameters

port

[int, optional] Localhost port to deploy the Geometry service on or the the Modeler interface to connect to (if it is already deployed). By default, the value is the one for the DEFAULT_PORT connection parameter.

connect_to_existing_service

[bool, default: True] Whether the Modeler interface should connect to a Geometry service already deployed at the specified port.

restart_if_existing_service

[bool, default: False] Whether the Geometry service (which is already running) should be restarted when attempting connection.

name

[str or None, default: None] Name of the Docker container to deploy. The default is None, in which case Docker assigns it a random name.

image

[GeometryContainers or None, default: None] The Geometry service Docker image to deploy. The default is None, in which case the LocalDockerInstance class identifies the OS of your Docker engine and deploys the latest version of the Geometry service for that OS.

Overview

Properties

<i>container</i>	Docker container object that hosts the deployed Geometry service.
<i>existed_previously</i>	Flag indicating whether the container previously existed.

Attributes

<code>__DOCKER_CLIENT__</code>	Docker client class variable.
--------------------------------	-------------------------------

Static methods

<code>docker_client</code>	Get the initialized <code>__DOCKER_CLIENT__</code> object.
<code>is_docker_installed</code>	Check a local installation of Docker engine is available and running.

Import detail

```
from ansys.geometry.core.connection.docker_instance import LocalDockInstance
```

Property detail

property LocalDockInstance.**container**: `docker.models.containers.Container`

Docker container object that hosts the deployed Geometry service.

property LocalDockInstance.**existed_previously**: `bool`

Flag indicating whether the container previously existed.

Returns `False` if the Geometry service was effectively deployed by this class or `True` if it already existed.

Attribute detail

LocalDockInstance.**__DOCKER_CLIENT__**: `docker.client.DockerClient = None`

Docker client class variable.

Notes

The default is `None`, in which case lazy initialization is used. `__DOCKER_CLIENT__` is a class variable, meaning that it is the same variable for all instances of this class.

Method detail

static LocalDockInstance.**docker_client**() → `docker.client.DockerClient`

Get the initialized `__DOCKER_CLIENT__` object.

Returns

`DockerClient`

Initialized Docker client.

Notes

The `LocalDockInstance` class performs a lazy initialization of the `__DOCKER_CLIENT__` class variable.

static LocalDockInstance.**is_docker_installed**() → `bool`

Check a local installation of Docker engine is available and running.

Returns

`bool`

True if Docker engine is available and running, `False` otherwise.

GeometryContainers

class `ansys.geometry.core.connection.docker_instance.GeometryContainers(*args, **kwargs)`

Bases: `enum.Enum`

Provides an enum holding the available Geometry services.

Overview

Attributes

<code>CORE_WINDOWS_LATEST</code>
<code>CORE_LINUX_LATEST</code>
<code>CORE_WINDOWS_LATEST_UNSTABLE</code>
<code>CORE_LINUX_LATEST_UNSTABLE</code>
<code>WINDOWS_LATEST</code>
<code>WINDOWS_LATEST_UNSTABLE</code>
<code>WINDOWS_24_1</code>
<code>WINDOWS_24_2</code>
<code>WINDOWS_25_1</code>
<code>WINDOWS_25_2</code>
<code>CORE_WINDOWS_25_2</code>
<code>CORE_LINUX_25_2</code>

Import detail

```
from ansys.geometry.core.connection.docker_instance import GeometryContainers
```

Attribute detail

`GeometryContainers.CORE_WINDOWS_LATEST = (0, 'windows', 'core-windows-latest')`

`GeometryContainers.CORE_LINUX_LATEST = (1, 'linux', 'core-linux-latest')`

`GeometryContainers.CORE_WINDOWS_LATEST_UNSTABLE = (2, 'windows', 'core-windows-latest-unstable')`

`GeometryContainers.CORE_LINUX_LATEST_UNSTABLE = (3, 'linux', 'core-linux-latest-unstable')`

`GeometryContainers.WINDOWS_LATEST = (4, 'windows', 'windows-latest')`

`GeometryContainers.WINDOWS_LATEST_UNSTABLE = (5, 'windows', 'windows-latest-unstable')`

`GeometryContainers.WINDOWS_24_1 = (6, 'windows', 'windows-24.1')`

`GeometryContainers.WINDOWS_24_2 = (7, 'windows', 'windows-24.2')`

`GeometryContainers.WINDOWS_25_1 = (8, 'windows', 'windows-25.1')`

`GeometryContainers.WINDOWS_25_2 = (9, 'windows', 'windows-25.2')`

`GeometryContainers.CORE_WINDOWS_25_2 = (10, 'windows', 'core-windows-25.2')`

`GeometryContainers.CORE_LINUX_25_2 = (11, 'linux', 'core-linux-25.2')`

Description

Module for connecting to a local Geometry Service Docker container.

Module detail

`docker_instance.get_geometry_container_type(instance: LocalDockerInstance) → GeometryContainers | None`

Provide back the GeometryContainers value.

Parameters

instance

[LocalDockerInstance] The LocalDockerInstance object.

Returns

GeometryContainers or None

The GeometryContainer value corresponding to the previous image or None if not match.

Notes

This method returns the first hit on the available tags.

The launcher.py module

Summary

Functions

<code>launch_modeler</code>	Start the Modeler interface for PyAnsys Geometry.
<code>launch_remote_modeler</code>	Start the Geometry service remotely using the PIM API.
<code>launch_docker_modeler</code>	Start the Geometry service locally using Docker.
<code>launch_modeler_with_discovery_and_pimlig</code>	Start Ansys Discovery remotely using the PIM API.
<code>launch_modeler_with_geometry_service_and</code>	Start the Geometry service remotely using the PIM API.
<code>launch_modeler_with_spaceclaim_and_pimli</code>	Start Ansys SpaceClaim remotely using the PIM API.
<code>launch_modeler_with_geometry_service</code>	Start the Geometry service locally using the ProductInstance class.
<code>launch_modeler_with_discovery</code>	Start Ansys Discovery locally using the ProductInstance class.
<code>launch_modeler_with_spaceclaim</code>	Start Ansys SpaceClaim locally using the ProductInstance class.
<code>launch_modeler_with_core_service</code>	Start the Geometry Core service locally using the ProductInstance class.

Description

Module for connecting to instances of the Geometry service.

Module detail

`launcher.launch_modeler(mode: str = None, **kwargs: dict | None) → ansys.geometry.core.modeler.Modeler`

Start the Modeler interface for PyAnsys Geometry.

Parameters

mode

[*str*, default: *None*] Mode in which to launch the `Modeler` service. The default is *None*, in which case the method tries to determine the mode automatically. The possible values are:

- "pypim": Launches the `Modeler` service remotely using the PIM API.
- "docker": Launches the `Modeler` service locally using Docker.
- "geometry_service": Launches the `Modeler` service locally using the Ansys Geometry Service.
- "spaceclaim": Launches the `Modeler` service locally using Ansys SpaceClaim.
- "discovery": Launches the `Modeler` service locally using Ansys Discovery.

****kwargs**

[*dict*, default: *None*] Keyword arguments for the launching methods. For allowable keyword arguments, see the corresponding methods for each mode:

- For "pypim" mode, see the `launch_remote_modeler()` method.
- For "docker" mode, see the `launch_docker_modeler()` method.
- For "geometry_service" mode, see the `launch_modeler_with_geometry_service()` method.
- For "core_service" mode, see the `launch_modeler_with_core_service()` method.
- For "spaceclaim" mode, see the `launch_modeler_with_spaceclaim()` method.
- For "discovery" mode, see the `launch_modeler_with_discovery()` method.

Returns

`ansys.geometry.core.modeler.Modeler`
Pythonic interface for geometry modeling.

Examples

Launch the Geometry service.

```
>>> from ansys.geometry.core import launch_modeler
>>> modeler = launch_modeler()
```

```
launcher.launch_remote_modeler(platform: str = 'windows', version: str | None = None, client_log_level: int = logging.INFO, client_log_file: str | None = None, **kwargs: dict | None)
→ ansys.geometry.core.modeler.Modeler
```

Start the Geometry service remotely using the PIM API.

When calling this method, you must ensure that you are in an environment where `PyPIM` is configured. You can use the `pypim.is_configured` method to check if it is configured.

Parameters**platform**

[*str*, default: *None*] **Specific for Ansys Lab.** The platform option for the Geometry service. The default is "windows". This parameter is used to specify the operating system on which the Geometry service will run. The possible values are:

- "windows": The Geometry service runs on a Windows machine.
- "linux": The Geometry service runs on a Linux machine.

version

[*str*, default: *None*] Version of the Geometry service to run in the three-digit format. For example, “241”. If you do not specify the version, the server chooses the version.

client_log_level

[*int*, default: `logging.INFO`] Log level for the client. The default is `logging.INFO`.

client_log_file

[*str*, default: *None*] Path to the log file for the client. The default is *None*, in which case the client logs to the console.

****kwargs**

[*dict*, default: *None*] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns***ansys.geometry.core.modeler.Modeler***

Instance of the Geometry service.

```
launcher.launch_docker_modeler(port: int = DEFAULT_PORT, connect_to_existing_service: bool = True,
                               restart_if_existing_service: bool = False, name: str | None = None, image:
                               ansys.geometry.core.connection.docker_instance.GeometryContainers |
                               None = None, client_log_level: int = logging.INFO, client_log_file: str |
                               None = None, **kwargs: dict | None) →
                               ansys.geometry.core.modeler.Modeler
```

Start the Geometry service locally using Docker.

When calling this method, a Geometry service (as a local Docker container) is started. By default, if a container with the Geometry service already exists at the given port, it connects to it. Otherwise, it tries to launch its own service.

Parameters**port**

[*int*, optional] Localhost port to deploy the Geometry service on or the the *Modeler* interface to connect to (if it is already deployed). By default, the value is the one for the `DEFAULT_PORT` connection parameter.

connect_to_existing_service

[*bool*, default: *True*] Whether the *Modeler* interface should connect to a Geometry service already deployed at the specified port.

restart_if_existing_service

[*bool*, default: *False*] Whether the Geometry service (which is already running) should be restarted when attempting connection.

name

[*str*, default: *None*] Name of the Docker container to deploy. The default is *None*, in which case Docker assigns it a random name.

image

[*GeometryContainers*, default: *None*] The Geometry service Docker image to deploy. The default is *None*, in which case the `LocalDockerInstance` class identifies the OS of your Docker engine and deploys the latest version of the Geometry service for that OS.

client_log_level

[*int*, default: `logging.INFO`] Log level for the client. The default is `logging.INFO`.

client_log_file

[*str*, default: *None*] Path to the log file for the client. The default is *None*, in which case the client logs to the console.

****kwargs**

[*dict*, default: *None*] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns**Modeler**

Instance of the Geometry service.

```
launcher.launch_modeler_with_discovery_and_pimlight(version: str | None = None, client_log_level: int = logging.INFO, client_log_file: str | None = None, **kwargs: dict | None) → ansys.geometry.core.modeler.Modeler
```

Start Ansys Discovery remotely using the PIM API.

When calling this method, you must ensure that you are in an environment where **PyPIM** is configured. You can use the `pypim.is_configured` method to check if it is configured.

Parameters**version**

[*str*, default: *None*] Version of Discovery to run in the three-digit format. For example, “241”. If you do not specify the version, the server chooses the version.

client_log_level

[*int*, default: `logging.INFO`] Log level for the client. The default is `logging.INFO`.

client_log_file

[*str*, default: *None*] Path to the log file for the client. The default is *None*, in which case the client logs to the console.

****kwargs**

[*dict*, default: *None*] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns**ansys.geometry.core.modeler.Modeler**

Instance of Modeler.

```
launcher.launch_modeler_with_geometry_service_and_pimlight(version: str | None = None, client_log_level: int = logging.INFO, client_log_file: str | None = None, **kwargs: dict | None) → ansys.geometry.core.modeler.Modeler
```

Start the Geometry service remotely using the PIM API.

When calling this method, you must ensure that you are in an environment where **PyPIM** is configured. You can use the `pypim.is_configured` method to check if it is configured.

Parameters**version**

[*str*, default: *None*] Version of the Geometry service to run in the three-digit format. For example, “241”. If you do not specify the version, the server chooses the version.

client_log_level

[*int*, default: `logging.INFO`] Log level for the client. The default is `logging.INFO`.

client_log_file

[*str*, default: *None*] Path to the log file for the client. The default is *None*, in which case the client logs to the console.

****kwargs**

[*dict*, default: *None*] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns

ansys.geometry.core.modeler.Modeler

Instance of *Modeler*.

`launcher.launch_modeler_with_spaceclaim_and_pimlight`(*version: str | None = None, client_log_level: int = logging.INFO, client_log_file: str | None = None, **kwargs: dict | None*) → *ansys.geometry.core.modeler.Modeler*

Start Ansys SpaceClaim remotely using the PIM API.

When calling this method, you must ensure that you are in an environment where *PyPIM* is configured. You can use the `pypim.is_configured` method to check if it is configured.

Parameters

version

[*str*, default: *None*] Version of SpaceClaim to run in the three-digit format. For example, “241”. If you do not specify the version, the server chooses the version.

client_log_level

[*int*, default: `logging.INFO`] Log level for the client. The default is `logging.INFO`.

client_log_file

[*str*, default: *None*] Path to the log file for the client. The default is *None*, in which case the client logs to the console.

****kwargs**

[*dict*, default: *None*] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns

ansys.geometry.core.modeler.Modeler

Instance of *Modeler*.

`launcher.launch_modeler_with_geometry_service`(*product_version: int = None, host: str = 'localhost', port: int = None, enable_trace: bool = False, timeout: int = 60, server_log_level: int = 2, client_log_level: int = logging.INFO, server_logs_folder: str = None, client_log_file: str = None, log_level: int = None, logs_folder: str = None, **kwargs: dict | None*) → *ansys.geometry.core.modeler.Modeler*

Start the Geometry service locally using the *ProductInstance* class.

When calling this method, a standalone Geometry service is started. By default, if an endpoint is specified (by defining *host* and *port* parameters) but the endpoint is not available, the startup will fail. Otherwise, it will try to launch its own service.

Parameters

product_version: int, optional

The product version to be started. Goes from v24.1 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

Ansys products versions and their corresponding int values:

- 241 : Ansys 24R1
- 242 : Ansys 24R2

host: str, optional

IP address at which the Geometry service will be deployed. By default, its value will be localhost.

port

[int, optional] Port at which the Geometry service will be deployed. By default, its value will be None.

enable_trace

[bool, optional] Boolean enabling the logs trace on the Geometry service console window. By default its value is False.

timeout

[int, optional] Timeout for starting the backend startup process. The default is 60.

server_log_level

[int, optional]

Backend's log level from 0 to 3:

0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

client_log_level

[int, optional] Logging level to apply to the client. By default, INFO level is used. Use the logging module's levels: DEBUG, INFO, WARNING, ERROR, CRITICAL.

server_logs_folder

[str, optional] Sets the backend's logs folder path. If nothing is defined, the backend will use its default path.

client_log_file

[str, optional] Sets the client's log file path. If nothing is defined, the client will log to the console.

log_level

[int, optional] DEPRECATED. Use server_log_level instead.

logs_folder

[str, optional] DEPRECATED. Use server_logs_folder instead.

****kwargs**

[dict, default: None] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

If the specified endpoint is already in use, a connection error will be raised.

SystemError

If there is not an Ansys product 24.1 version or later installed a SystemError will be raised.

Examples

Starting a geometry service with the default parameters and getting back a Modeler object:

```
>>> from ansys.geometry.core import launch_modeler_with_geometry_service
>>> modeler = launch_modeler_with_geometry_service()
```

Starting a geometry service, on address 10.171.22.44, port 5001, with chatty logs, traces enabled and a 300 seconds timeout:

```
>>> from ansys.geometry.core import launch_modeler_with_geometry_service
>>> modeler = launch_modeler_with_geometry_service(host="10.171.22.44",
port=5001,
enable_trace= True,
timeout=300,
server_log_level=0)
```

```
launcher.launch_modeler_with_discovery(product_version: int = None, host: str = 'localhost', port: int =
None, api_version:
ansys.geometry.core.connection.backend.ApiVersions =
ApiVersions.LATEST, timeout: int = 150, manifest_path: str =
None, hidden: bool = False, server_log_level: int = 2,
client_log_level: int = logging.INFO, client_log_file: str = None,
log_level: int = None, **kwargs: dict | None)
```

Start Ansys Discovery locally using the ProductInstance class.

When calling this method, a standalone Discovery session is started. By default, if an endpoint is specified (by defining *host* and *port* parameters) but the endpoint is not available, the startup will fail. Otherwise, it will try to launch its own service.

Parameters**product_version: int, optional**

The product version to be started. Goes from v24.1 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

Ansys products versions and their corresponding int values:

- 241 : Ansys 24R1
- 242 : Ansys 24R2

host: str, optional

IP address at which the Discovery session will be deployed. By default, its value will be localhost.

port

[int, optional] Port at which the Geometry service will be deployed. By default, its value will be None.

api_version: ApiVersions, optional

The backend's API version to be used at runtime. Goes from API v21 to the latest. Default is ApiVersions.LATEST.

timeout

[int, optional] Timeout for starting the backend startup process. The default is 150.

manifest_path

[`str`, optional] Used to specify a manifest file path for the `ApiServerAddin`. This way, it is possible to run an `ApiServerAddin` from a version an older product version.

hidden

[starts the product hiding its UI. Default is `False`.]

server_log_level

[`int`, optional]

Backend's log level from 0 to 3:

0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

client_log_level

[`int`, optional] Logging level to apply to the client. By default, INFO level is used. Use the logging module's levels: DEBUG, INFO, WARNING, ERROR, CRITICAL.

client_log_file

[`str`, optional] Sets the client's log file path. If nothing is defined, the client will log to the console.

log_level

[`int`, optional] DEPRECATED. Use `server_log_level` instead.

****kwargs**

[`dict`, default: `None`] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

If the specified endpoint is already in use, a connection error will be raised.

SystemError:

If there is not an Ansys product 24.1 version or later installed or if a specific product's version is requested but not installed locally then a `SystemError` will be raised.

Examples

Starting an Ansys Discovery session with the default parameters and getting back a `Modeler` object:

```
>>> from ansys.geometry.core import launch_modeler_with_discovery
>>> modeler = launch_modeler_with_discovery()
```

Starting an Ansys Discovery V 24.1 session, on address `10.171.22.44`, port `5001`, with chatty logs, using API `v231` and a `300` seconds timeout:

```
>>> from ansys.geometry.core import launch_modeler_with_discovery
>>> modeler = launch_modeler_with_discovery(product_version = 241,
host="10.171.22.44",
port=5001,
api_version= 231,
timeout=300,
server_log_level=0)
```

```
launcher.launch_modeler_with_spaceclaim(product_version: int = None, host: str = 'localhost', port: int =
None, api_version:
ansys.geometry.core.connection.backend.ApiVersions =
ApiVersions.LATEST, timeout: int = 150, manifest_path: str =
None, hidden: bool = False, server_log_level: int = 2,
client_log_level: int = logging.INFO, client_log_file: str =
None, log_level: int = None, **kwargs: dict | None)
```

Start Ansys SpaceClaim locally using the ProductInstance class.

When calling this method, a standalone SpaceClaim session is started. By default, if an endpoint is specified (by defining *host* and *port* parameters) but the endpoint is not available, the startup will fail. Otherwise, it will try to launch its own service.

Parameters

product_version: int, optional

The product version to be started. Goes from v24.1 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

Ansys products versions and their corresponding int values:

- 241 : Ansys 24R1
- 242 : Ansys 24R2

host: str, optional

IP address at which the SpaceClaim session will be deployed. By default, its value will be localhost.

port

[int, optional] Port at which the Geometry service will be deployed. By default, its value will be None.

api_version: ApiVersions, optional

The backend's API version to be used at runtime. Goes from API v21 to the latest. Default is ApiVersions.LATEST.

timeout

[int, optional] Timeout for starting the backend startup process. The default is 150.

manifest_path

[str, optional] Used to specify a manifest file path for the ApiServerAddin. This way, it is possible to run an ApiServerAddin from a version an older product version.

hidden

[starts the product hiding its UI. Default is False.]

server_log_level

[int, optional]

Backend's log level from 0 to 3:

0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

client_log_level

[int, optional] Logging level to apply to the client. By default, INFO level is used. Use the logging module's levels: DEBUG, INFO, WARNING, ERROR, CRITICAL.

client_log_file

[str, optional] Sets the client's log file path. If nothing is defined, the client will log to the console.

log_level

[int, optional] DEPRECATED. Use server_log_level instead.

****kwargs**

[dict, default: None] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

If the specified endpoint is already in use, a connection error will be raised.

SystemError

If there is not an Ansys product 24.1 version or later installed or if a specific product's version is requested but not installed locally then a SystemError will be raised.

Examples

Starting an Ansys SpaceClaim session with the default parameters and get back a Modeler object:

```
>>> from ansys.geometry.core import launch_modeler_with_spaceclaim
>>> modeler = launch_modeler_with_spaceclaim()
```

Starting an Ansys SpaceClaim V 24.1 session, on address 10.171.22.44, port 5001, with chatty logs, using API v231 and a 300 seconds timeout:

```
>>> from ansys.geometry.core import launch_modeler_with_spaceclaim
>>> modeler = launch_modeler_with_spaceclaim(product_version = 241,
host="10.171.22.44",
port=5001,
api_version= 231,
timeout=300,
server_log_level=0)
```

```
launcher.launch_modeler_with_core_service(product_version: int = None, host: str = 'localhost', port: int = None, enable_trace: bool = False, timeout: int = 60, server_log_level: int = 2, client_log_level: int = logging.INFO, server_logs_folder: str = None, client_log_file: str = None, **kwargs: dict | None) → ansys.geometry.core.modeler.Modeler
```

Start the Geometry Core service locally using the ProductInstance class.

When calling this method, a standalone Geometry Core service is started. By default, if an endpoint is specified (by defining *host* and *port* parameters) but the endpoint is not available, the startup will fail. Otherwise, it will try to launch its own service.

Parameters**product_version: int, optional**

The product version to be started. Goes from v25.2 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

Ansys products versions and their corresponding int values:

- 252 : Ansys 25R2

- 261 : Ansys 26R1

host: str, optional

IP address at which the service will be deployed. By default, its value will be localhost.

port

[int, optional] Port at which the service will be deployed. By default, its value will be None.

enable_trace

[bool, optional] Boolean enabling the logs trace on the service console window. By default its value is False.

timeout

[int, optional] Timeout for starting the backend startup process. The default is 60.

server_log_level

[int, optional]

Backend's log level from 0 to 3:

0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

client_log_level

[int, optional] Logging level to apply to the client. By default, INFO level is used. Use the logging module's levels: DEBUG, INFO, WARNING, ERROR, CRITICAL.

server_logs_folder

[str, optional] Sets the backend's logs folder path. If nothing is defined, the backend will use its default path.

client_log_file

[str, optional] Sets the client's log file path. If nothing is defined, the client will log to the console.

****kwargs**

[dict, default: None] Placeholder to prevent errors when passing additional arguments that are not compatible with this method.

Returns

Modeler

Instance of the Geometry Core service.

Raises

ConnectionError

If the specified endpoint is already in use, a connection error will be raised.

SystemError

If there is not an Ansys product 25.2 version or later installed a SystemError will be raised.

Examples

Starting a geometry core service with the default parameters and getting back a Modeler object:

```
>>> from ansys.geometry.core import launch_modeler_with_core_service
>>> modeler = launch_modeler_with_core_service()
```

Starting a geometry service, on address 10.171.22.44, port 5001, with chatty logs, traces enabled and a 300 seconds timeout:

```
>>> from ansys.geometry.core import launch_modeler_with_core_service
>>> modeler = launch_modeler_with_core_service(host="10.171.22.44",
port=5001,
enable_trace= True,
timeout=300,
server_log_level=0)
```

The product_instance.py module

Summary

Classes

<i>ProductInstance</i>	ProductInstance class.
------------------------	------------------------

Functions

<i>prepare_and_start_backend</i>	Start the requested service locally using the ProductInstance class.
<i>get_available_port</i>	Return an available port to be used.

Constants

<i>WINDOWS_GEOMETRY_SERVICE_FOLDE</i>	Default Geometry Service's folder name into the unified installer (DMS).
<i>CORE_GEOMETRY_SERVICE_FOLDER</i>	Default Geometry Service's folder name into the unified installer (Core Service).
<i>DISCOVERY_FOLDER</i>	Default Discovery's folder name into the unified installer.
<i>SPACECLAIM_FOLDER</i>	Default SpaceClaim's folder name into the unified installer.
<i>ADDINS_SUBFOLDER</i>	Default global Addins's folder name into the unified installer.
<i>BACKEND_SUBFOLDER</i>	Default backend's folder name into the ADDINS_SUBFOLDER folder.
<i>MANIFEST_FILENAME</i>	Default backend's add-in filename.
<i>GEOMETRY_SERVICE_EXE</i>	The Windows Geometry Service's filename (DMS).
<i>CORE_GEOMETRY_SERVICE_EXE</i>	The Windows Geometry Service's filename (Core Service).
<i>DISCOVERY_EXE</i>	The Ansys Discovery's filename.
<i>SPACECLAIM_EXE</i>	The Ansys SpaceClaim's filename.
<i>BACKEND_LOG_LEVEL_VARIABLE</i>	The backend's log level environment variable for local start.
<i>BACKEND_TRACE_VARIABLE</i>	The backend's enable trace environment variable for local start.
<i>BACKEND_HOST_VARIABLE</i>	The backend's ip address environment variable for local start.
<i>BACKEND_PORT_VARIABLE</i>	The backend's port number environment variable for local start.
<i>BACKEND_LOGS_FOLDER_VARIABLE</i>	The backend's logs folder path to be used.
<i>BACKEND_API_VERSION_VARIABLE</i>	The backend's api version environment variable for local start.
<i>BACKEND_SPACECLAIM_OPTIONS</i>	The additional argument for local Ansys Discovery start.
<i>BACKEND_ADDIN_MANIFEST_ARGUMEN</i>	The argument to specify the backend's add-in manifest file's path.
<i>BACKEND_SPACECLAIM_HIDDEN</i>	The argument to hide SpaceClaim's UI on the backend.
<i>BACKEND_SPACECLAIM_HIDDEN_ENVV</i>	SpaceClaim hidden backend's environment variable key.
<i>BACKEND_SPACECLAIM_HIDDEN_ENVV</i>	SpaceClaim hidden backend's environment variable value.
<i>BACKEND_DISCOVERY_HIDDEN</i>	The argument to hide Discovery's UI on the backend.
<i>BACKEND_SPLASH_OFF</i>	The argument to specify the backend's add-in manifest file's path.
<i>ANSYS_GEOMETRY_SERVICE_ROOT</i>	Local Geometry Service install location. This is for GeometryService and CoreGeometryService.

ProductInstance

class `ansys.geometry.core.connection.product_instance.ProductInstance(pid: int)`

ProductInstance class.

This class is used as a handle for a local session of Ansys Product's backend: Discovery, Windows Geometry Service or SpaceClaim.

Parameters

pid

[int] The local instance's process identifier. This allows to keep track of the process and close it if need be.

Overview

Methods

<code>close</code> Close the process associated to the pid.

Import detail

```
from ansys.geometry.core.connection.product_instance import ProductInstance
```

Method detail

`ProductInstance.close()` → `bool`

Close the process associated to the pid.

Description

Module containing the ProductInstance class.

Module detail

`product_instance.prepare_and_start_backend(backend_type: ansys.geometry.core.connection.backend.BackendType, product_version: int = None, host: str = 'localhost', port: int = None, enable_trace: bool = False, api_version: ansys.geometry.core.connection.backend.ApiVersions = ApiVersions.LATEST, timeout: int = 150, manifest_path: str = None, hidden: bool = False, server_log_level: int = 2, client_log_level: int = logging.INFO, server_logs_folder: str = None, client_log_file: str = None, specific_minimum_version: int = None, log_level: int = None, logs_folder: str = None) → ansys.geometry.core.modeler.Modeler`

Start the requested service locally using the ProductInstance class.

When calling this method, a standalone service or product session is started. By default, if an endpoint is specified (by defining `host` and `port` parameters) but the endpoint is not available, the startup will fail. Otherwise, it will try to launch its own service.

Parameters

product_version: ``int``, optional

The product version to be started. Goes from v24.1 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

host: str, optional

IP address at which the Geometry service will be deployed. By default, its value will be localhost.

port

[int, optional] Port at which the Geometry service will be deployed. By default, its value will be None.

enable_trace

[bool, optional] Boolean enabling the logs trace on the Geometry service console window. By default its value is False.

api_version: ``ApiVersions``, optional

The backend's API version to be used at runtime. Goes from API v21 to the latest. Default is ApiVersions.LATEST.

timeout

[int, optional] Timeout for starting the backend startup process. The default is 150.

manifest_path

[str, optional] Used to specify a manifest file path for the ApiServerAddin. This way, it is possible to run an ApiServerAddin from a version an older product version. Only applicable for Ansys Discovery and Ansys SpaceClaim.

hidden

[starts the product hiding its UI. Default is False.]

server_log_level

[int, optional]

Backend's log level from 0 to 3:

0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

client_log_level

[int, optional] Logging level to apply to the client. By default, INFO level is used. Use the logging module's levels: DEBUG, INFO, WARNING, ERROR, CRITICAL.

server_logs_folder

[str, optional] Sets the backend's logs folder path. If nothing is defined, the backend will use its default path.

client_log_file

[str, optional] Sets the client's log file path. If nothing is defined, the client will log to the console.

specific_minimum_version

[int, optional] Sets a specific minimum version to be checked. If this is not defined, the minimum version will be set to 24.1.0.

log_level

[int, optional] DEPRECATED. Use server_log_level instead.

logs_folder

[str, optional] DEPRECATED. Use server_logs_folder instead.

Returns

Modeler

Instance of the Geometry service.

Raises

ConnectionError

If the specified endpoint is already in use, a connection error will be raised.

SystemError

If there is not an Ansys product 24.1 version or later installed or if a specific product's version is requested but not installed locally then a SystemError will be raised.

`product_instance.get_available_port() → int`

Return an available port to be used.

Returns

int

The available port.

`product_instance.WINDOWS_GEOMETRY_SERVICE_FOLDER = 'GeometryService'`

Default Geometry Service's folder name into the unified installer (DMS).

`product_instance.CORE_GEOMETRY_SERVICE_FOLDER = 'CoreGeometryService'`

Default Geometry Service's folder name into the unified installer (Core Service).

`product_instance.DISCOVERY_FOLDER = 'Discovery'`

Default Discovery's folder name into the unified installer.

`product_instance.SPACECLAIM_FOLDER = 'scdm'`

Default SpaceClaim's folder name into the unified installer.

`product_instance.ADDINS_SUBFOLDER = 'Addins'`

Default global Addins's folder name into the unified installer.

`product_instance.BACKEND_SUBFOLDER = 'ApiServer'`

Default backend's folder name into the ADDINS_SUBFOLDER folder.

`product_instance.MANIFEST_FILENAME = 'Presentation.ApiServerAddIn.Manifest.xml'`

Default backend's add-in filename.

To be used only for local start of Ansys Discovery or Ansys SpaceClaim.

`product_instance.GEOMETRY_SERVICE_EXE = 'Presentation.ApiServerDMS.exe'`

The Windows Geometry Service's filename (DMS).

`product_instance.CORE_GEOMETRY_SERVICE_EXE = 'Presentation.ApiServerLinux.exe'`

The Windows Geometry Service's filename (Core Service).

`product_instance.DISCOVERY_EXE = 'Discovery.exe'`

The Ansys Discovery's filename.

`product_instance.SPACECLAIM_EXE = 'SpaceClaim.exe'`

The Ansys SpaceClaim's filename.

`product_instance.BACKEND_LOG_LEVEL_VARIABLE = 'LOG_LEVEL'`

The backend's log level environment variable for local start.

`product_instance.BACKEND_TRACE_VARIABLE = 'ENABLE_TRACE'`

The backend's enable trace environment variable for local start.

`product_instance.BACKEND_HOST_VARIABLE = 'API_ADDRESS'`

The backend's ip address environment variable for local start.

`product_instance.BACKEND_PORT_VARIABLE = 'API_PORT'`

The backend's port number environment variable for local start.

`product_instance.BACKEND_LOGS_FOLDER_VARIABLE = 'ANS_DSCO_REMOTE_LOGS_FOLDER'`

The backend's logs folder path to be used.

Only applicable to the Ansys Geometry Service.

`product_instance.BACKEND_API_VERSION_VARIABLE = 'API_VERSION'`

The backend's api version environment variable for local start.

To be used only with Ansys Discovery and Ansys SpaceClaim.

`product_instance.BACKEND_SPACECLAIM_OPTIONS = '--spaceclaim-options'`

The additional argument for local Ansys Discovery start.

To be used only with Ansys Discovery.

`product_instance.BACKEND_ADDIN_MANIFEST_ARGUMENT = '/ADDINMANIFESTFILE='`

The argument to specify the backend's add-in manifest file's path.

To be used only with Ansys Discovery and Ansys SpaceClaim.

`product_instance.BACKEND_SPACECLAIM_HIDDEN = '/Headless=True'`

The argument to hide SpaceClaim's UI on the backend.

To be used only with Ansys SpaceClaim.

`product_instance.BACKEND_SPACECLAIM_HIDDEN_ENVVAR_KEY = 'SPACECLAIM_MODE'`

SpaceClaim hidden backend's environment variable key.

To be used only with Ansys SpaceClaim.

`product_instance.BACKEND_SPACECLAIM_HIDDEN_ENVVAR_VALUE = '2'`

SpaceClaim hidden backend's environment variable value.

To be used only with Ansys SpaceClaim.

`product_instance.BACKEND_DISCOVERY_HIDDEN = '--hidden'`

The argument to hide Discovery's UI on the backend.

To be used only with Ansys Discovery.

`product_instance.BACKEND_SPLASH_OFF = '/Splash=False'`

The argument to specify the backend's add-in manifest file's path.

To be used only with Ansys Discovery and Ansys SpaceClaim.

`product_instance.ANSYS_GEOMETRY_SERVICE_ROOT = 'ANSYS_GEOMETRY_SERVICE_ROOT'`

Local Geometry Service install location. This is for GeometryService and CoreGeometryService.

The `validate.py` module

Summary

Functions

<code>validate</code>	Create a client using the default settings and validate it.
-----------------------	---

Description

Module to perform a connection validation check.

The method in this module is only used for testing the default Docker service on GitHub and can safely be skipped within testing.

This command shows how this method is typically used:

```
python -c "from ansys.geometry.core.connection import validate; validate()"
```

Module detail

`validate.validate(*args, **kwargs)`

Create a client using the default settings and validate it.

Description

PyAnsys Geometry connection subpackage.

The designer package

Summary

Submodules

<i>beam</i>	Provides for creating and managing a beam.
<i>body</i>	Provides for managing a body.
<i>component</i>	Provides for managing components.
<i>coordinate_system</i>	Provides for managing a user-defined coordinate system.
<i>design</i>	Provides for managing designs.
<i>designpoint</i>	Module for creating and managing design points.
<i>edge</i>	Module for managing an edge.
<i>face</i>	Module for managing a face.
<i>geometry_commands</i>	Provides tools for pulling geometry.
<i>part</i>	Module providing fundamental data of an assembly.
<i>selection</i>	Module for creating a named selection.

The beam.py module

Summary

Classes

<i>BeamProfile</i>	Represents a single beam profile organized within the design assembly.
<i>BeamCircularProfile</i>	Represents a single circular beam profile.
<i>Beam</i>	Represents a simplified solid body with an assigned 2D cross-section.

BeamProfile

`class ansys.geometry.core.designer.beam.BeamProfile(id: str, name: str)`

Represents a single beam profile organized within the design assembly.

This profile synchronizes to a design within a supporting Geometry service instance.

Parameters

- id**
[str] Server-defined ID for the beam profile.
- name**
[str] User-defined label for the beam profile.

Notes

BeamProfile objects are expected to be created from the Design object. This means that you are not expected to instantiate your own BeamProfile object. You should call the specific Design API for the BeamProfile desired.

Overview

Properties

<i>id</i>	ID of the beam profile.
<i>name</i>	Name of the beam profile.

Import detail

```
from ansys.geometry.core.designer.beam import BeamProfile
```

Property detail

property BeamProfile.id: str

ID of the beam profile.

property BeamProfile.name: str

Name of the beam profile.

BeamCircularProfile

```
class ansys.geometry.core.designer.beam.BeamCircularProfile(id: str, name: str, radius: an-
    sys.geometry.core.misc.measurements.Distance,
    center: an-
    sys.geometry.core.math.point.Point3D,
    direction_x: an-
    sys.geometry.core.math.vector.UnitVector3D,
    direction_y: an-
    sys.geometry.core.math.vector.UnitVector3D)
```

Bases: BeamProfile

Represents a single circular beam profile.

This profile synchronizes to a design within a supporting Geometry service instance.

Parameters

- id**
[str] Server-defined ID for the beam profile.
- name**
[str] User-defined label for the beam profile.

radius

[Distance] Radius of the circle.

center: Point3D

3D point representing the center of the circle.

direction_x: UnitVector3D

X-axis direction.

direction_y: UnitVector3D

Y-axis direction.

Notes

BeamProfile objects are expected to be created from the Design object. This means that you are not expected to instantiate your own BeamProfile object. You should call the specific Design API for the BeamProfile desired.

Overview**Properties**

<i>radius</i>	Radius of the circular beam profile.
<i>center</i>	Center of the circular beam profile.
<i>direction_x</i>	X-axis direction of the circular beam profile.
<i>direction_y</i>	Y-axis direction of the circular beam profile.

Special methods

<code>__repr__</code>	Represent the BeamCircularProfile as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.designer.beam import BeamCircularProfile
```

Property detail

property BeamCircularProfile.**radius**: *ansys.geometry.core.misc.measurements.Distance*

Radius of the circular beam profile.

property BeamCircularProfile.**center**: *ansys.geometry.core.math.point.Point3D*

Center of the circular beam profile.

property BeamCircularProfile.**direction_x**: *ansys.geometry.core.math.vector.UnitVector3D*

X-axis direction of the circular beam profile.

property BeamCircularProfile.**direction_y**: *ansys.geometry.core.math.vector.UnitVector3D*

Y-axis direction of the circular beam profile.

Method detail

`BeamCircularProfile.__repr__()` → `str`

Represent the `BeamCircularProfile` as a string.

Beam

```
class ansys.geometry.core.designer.beam.Beam(id: str, start: ansys.geometry.core.math.point.Point3D,
                                             end: ansys.geometry.core.math.point.Point3D, profile:
                                             BeamProfile, parent_component:
                                             ansys.geometry.core.designer.component.Component)
```

Represents a simplified solid body with an assigned 2D cross-section.

This body synchronizes to a design within a supporting Geometry service instance.

Parameters

id

[`str`] Server-defined ID for the body.

name

[`str`] User-defined label for the body.

start

[`Point3D`] Start of the beam line segment.

end

[`Point3D`] End of the beam line segment.

profile

[`BeamProfile`] Beam profile to use to create the beam.

parent_component

[`Component`] Parent component to nest the new beam under within the design assembly.

Overview

Properties

<code>id</code>	Service-defined ID of the beam.
<code>start</code>	Start of the beam line segment.
<code>end</code>	End of the beam line segment.
<code>profile</code>	Beam profile of the beam line segment.
<code>parent_component</code>	Component node that the beam is under.
<code>is_alive</code>	Flag indicating whether the beam is still alive on the server.

Special methods

```
__repr__ Represent the beam as a string.
```

Import detail

```
from ansys.geometry.core.designer.beam import Beam
```


Property detail

property `Beam.id: str`

Service-defined ID of the beam.

property `Beam.start: ansys.geometry.core.math.point.Point3D`

Start of the beam line segment.

property `Beam.end: ansys.geometry.core.math.point.Point3D`

End of the beam line segment.

property `Beam.profile: BeamProfile`

Beam profile of the beam line segment.

property `Beam.parent_component: ansys.geometry.core.designer.component.Component`

Component node that the beam is under.

property `Beam.is_alive: bool`

Flag indicating whether the beam is still alive on the server.

Method detail

`Beam.__repr__() → str`

Represent the beam as a string.

Description

Provides for creating and managing a beam.

The `body.py` module

Summary

Interfaces

<i>IBody</i>	Defines the common methods for a body, providing the abstract body interface.
--------------	---

Classes

<i>MasterBody</i>	Represents solids and surfaces organized within the design assembly.
<i>Body</i>	Represents solids and surfaces organized within the design assembly.

Enums

<i>MidSurfaceOffsetType</i>	Provides values for mid-surface offsets supported.
<i>CollisionType</i>	Provides values for collision types between bodies.
<i>FillStyle</i>	Provides values for fill styles supported.

Constants

__TEMPORARY_BOOL_OPS_FIX__

IBody

class `ansys.geometry.core.designer.body.IBody`

Bases: `abc.ABC`

Defines the common methods for a body, providing the abstract body interface.

Both the `MasterBody` class and `Body` class both inherit from the `IBody` class. All child classes must implement all abstract methods.

Overview

Abstract methods

<code>id</code>	Get the ID of the body as a string.
<code>name</code>	Get the name of the body.
<code>set_name</code>	Set the name of the body.
<code>fill_style</code>	Get the fill style of the body.
<code>set_fill_style</code>	Set the fill style of the body.
<code>is_suppressed</code>	Get the body suppression state.
<code>set_suppressed</code>	Set the body suppression state.
<code>color</code>	Get the color of the body.
<code>set_color</code>	Set the color of the body.
<code>faces</code>	Get a list of all faces within the body.
<code>edges</code>	Get a list of all edges within the body.
<code>is_alive</code>	Check if the body is still alive and has not been deleted.
<code>is_surface</code>	Check if the body is a planar body.
<code>surface_thickness</code>	Get the surface thickness of a surface body.
<code>surface_offset</code>	Get the surface offset type of a surface body.
<code>volume</code>	Calculate the volume of the body.
<code>material</code>	Get the assigned material of the body.
<code>assign_material</code>	Assign a material against the active design.
<code>get_assigned_material</code>	Get the assigned material of the body.
<code>add_midsurface_thickness</code>	Add a mid-surface thickness to a surface body.
<code>add_midsurface_offset</code>	Add a mid-surface offset to a surface body.
<code>imprint_curves</code>	Imprint all specified geometries onto specified faces of the body.
<code>project_curves</code>	Project all specified geometries onto the body.
<code>imprint_projected_curves</code>	Project and imprint specified geometries onto the body.
<code>translate</code>	Translate the body in a specified direction and distance.
<code>rotate</code>	Rotate the geometry body around the specified axis by a given angle.
<code>scale</code>	Scale the geometry body by the given value.
<code>map</code>	Map the geometry body to the new specified frame.
<code>mirror</code>	Mirror the geometry body across the specified plane.
<code>get_collision</code>	Get the collision state between bodies.
<code>copy</code>	Create a copy of the body under the specified parent.
<code>tessellate</code>	Tessellate the body and return the geometry as triangles.
<code>shell_body</code>	Shell the body to the thickness specified.
<code>remove_faces</code>	Shell by removing a given set of faces.

continues on next page

Table 1 – continued from previous page

<i>plot</i>	Plot the body.
-------------	----------------

Methods

<i>intersect</i>	Intersect two (or more) bodies.
<i>subtract</i>	Subtract two (or more) bodies.
<i>unite</i>	Unite two (or more) bodies.

Import detail

```
from ansys.geometry.core.designer.body import IBody
```

Method detail

abstract `IBody.id()` → `str`

Get the ID of the body as a string.

abstract `IBody.name()` → `str`

Get the name of the body.

abstract `IBody.set_name(str)` → `None`

Set the name of the body.

abstract `IBody.fill_style()` → `FillStyle`

Get the fill style of the body.

abstract `IBody.set_fill_style(fill_style: FillStyle)` → `None`

Set the fill style of the body.

abstract `IBody.is_suppressed()` → `bool`

Get the body suppression state.

abstract `IBody.set_suppressed(suppressed: bool)` → `None`

Set the body suppression state.

abstract `IBody.color()` → `str`

Get the color of the body.

abstract `IBody.set_color(color: str | tuple[float, float, float])` → `None`

Set the color of the body.

Parameters

color

[`str` | `tuple`[`float`, `float`, `float`]] Color to set the body to. This can be a string representing a color name or a tuple of RGB values in the range [0, 1] (RGBA) or [0, 255] (pure RGB).

abstract `IBody.faces()` → `list`[`ansys.geometry.core.designer.face.Face`]

Get a list of all faces within the body.

Returns

`list`[`Face`]

abstract `IBody.edges()` → `list[ansys.geometry.core.designer.edge.Edge]`

Get a list of all edges within the body.

Returns

`list[Edge]`

abstract `IBody.is_alive()` → `bool`

Check if the body is still alive and has not been deleted.

abstract `IBody.is_surface()` → `bool`

Check if the body is a planar body.

abstract `IBody.surface_thickness()` → `pint.Quantity | None`

Get the surface thickness of a surface body.

Notes

This method is only for surface-type bodies that have been assigned a surface thickness.

abstract `IBody.surface_offset()` → `MidSurfaceOffsetType | None`

Get the surface offset type of a surface body.

Notes

This method is only for surface-type bodies that have been assigned a surface offset.

abstract `IBody.volume()` → `pint.Quantity`

Calculate the volume of the body.

Notes

When dealing with a planar surface, a value of 0 is returned as a volume.

abstract `IBody.material()` → `ansys.geometry.core.materials.material.Material`

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

abstract `IBody.assign_material(material: ansys.geometry.core.materials.material.Material)` → `None`

Assign a material against the active design.

Parameters

material

[Material] Source material data.

abstract `IBody.get_assigned_material()` → `ansys.geometry.core.materials.material.Material`

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

abstract `IBody.add_midsurface_thickness`(*thickness*: *pint.Quantity*) → `None`

Add a mid-surface thickness to a surface body.

Parameters

thickness
[`Quantity`] Thickness to assign.

Notes

Only surface bodies are eligible for mid-surface thickness assignment.

abstract `IBody.add_midsurface_offset`(*offset*: `MidSurfaceOffsetType`) → `None`

Add a mid-surface offset to a surface body.

Parameters

offset_type
[`MidSurfaceOffsetType`] Surface offset to assign.

Notes

Only surface bodies are eligible for mid-surface offset assignment.

abstract `IBody.imprint_curves`(*faces*: *list*[`ansys.geometry.core.designer.face.Face`], *sketch*:
`ansys.geometry.core.sketch.sketch.Sketch`) →
`tuple`[*list*[`ansys.geometry.core.designer.edge.Edge`],
list[`ansys.geometry.core.designer.face.Face`]]

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: `list`[`Face`]
list of faces to imprint the curves of the sketch onto.

sketch: `Sketch`
All curves to imprint on the faces.

Returns

`tuple`[`list`[`Edge`], `list`[`Face`]]
All impacted edges and faces from the imprint operation.

abstract `IBody.project_curves`(*direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *sketch*:
`ansys.geometry.core.sketch.sketch.Sketch`, *closest_face*: `bool`,
only_one_curve: `bool = False`) →
`list`[`ansys.geometry.core.designer.face.Face`]]

Project all specified geometries onto the body.

Parameters

direction: `UnitVector3D`
Direction of the projection.

sketch: `Sketch`
All curves to project on the body.

closest_face: `bool`
Whether to target the closest face with the projection.

only_one_curve: bool, default: False

Whether to project only one curve of the entire sketch. When True, only one curve is projected.

Returns

`list[Face]`

All faces from the project curves operation.

Notes

The `only_one_curve` parameter allows you to optimize the server call because projecting curves is an expensive operation. This reduces the workload on the server side.

```
abstract IBody.imprint_projected_curves(direction: ansys.geometry.core.math.vector.UnitVector3D,
                                         sketch: ansys.geometry.core.sketch.sketch.Sketch, closest_face:
                                         bool, only_one_curve: bool = False) →
                                         list[ansys.geometry.core.designer.face.Face]
```

Project and imprint specified geometries onto the body.

This method combines the `project_curves()` and `imprint_curves()` method into one method. It has higher performance than calling them back-to-back when dealing with many curves. Because it is a specialized function, this method only returns the faces (and not the edges) from the imprint operation.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

Whether to project only one curve of the entire sketch. When True, only one curve is projected.

Returns

`list[Face]`

All imprinted faces from the operation.

Notes

The `only_one_curve` parameter allows you to optimize the server call because projecting curves is an expensive operation. This reduces the workload on the server side.

```
abstract IBody.translate(direction: ansys.geometry.core.math.vector.UnitVector3D, distance: pint.Quantity |
                        ansys.geometry.core.misc.measurements.Distance |
                        ansys.geometry.core.typing.Real) → None
```

Translate the body in a specified direction and distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Distance (magnitude) of the translation.

Returns**None**

abstract IBody.**rotate**(*axis_origin*: ansys.geometry.core.math.point.Point3D, *axis_direction*: ansys.geometry.core.math.vector.UnitVector3D, *angle*: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real) → None

Rotate the geometry body around the specified axis by a given angle.

Parameters**axis_origin: Point3D**

Origin of the rotational axis.

axis_direction: UnitVector3D

The axis of rotation.

angle: ~pint.Quantity | Angle | Real

Angle (magnitude) of the rotation.

Returns**None**

abstract IBody.**scale**(*value*: ansys.geometry.core.typing.Real) → None

Scale the geometry body by the given value.

Parameters**value: Real**

Value to scale the body by.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

abstract IBody.**map**(*frame*: ansys.geometry.core.math.frame.Frame) → None

Map the geometry body to the new specified frame.

Parameters**frame: Frame**

Structure defining the orientation of the body.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

abstract IBody.**mirror**(*plane*: ansys.geometry.core.math.plane.Plane) → None

Mirror the geometry body across the specified plane.

Parameters**plane: Plane**

Represents the mirror.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

abstract `IBody.get_collision(body: Body) → CollisionType`

Get the collision state between bodies.

Parameters

body: Body
Object that the collision state is checked with.

Returns

CollisionType
Enum that defines the collision state between bodies.

abstract `IBody.copy(parent: ansys.geometry.core.designer.component.Component, name: str = None) → Body`

Create a copy of the body under the specified parent.

Parameters

parent: Component
Parent component to place the new body under within the design assembly.

name: str
Name to give the new body.

Returns

Body
Copy of the body.

abstract `IBody.tessellate(merge: bool = False) → pyvista.PolyData | pyvista.MultiBlock`

Tessellate the body and return the geometry as triangles.

Parameters

merge
[bool, default: False] Whether to merge the body into a single mesh. When False (default), the number of triangles are preserved and only the topology is merged. When True, the individual faces of the tessellation are merged.

Returns

PolyData, MultiBlock
Merged `pyvista.PolyData` if `merge=True` or a composite dataset.

Examples

Extrude a box centered at the origin to create a rectangular body and tessellate it:

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> my_comp = design.add_component("my-comp")
>>> body = my_comp.extrude_sketch("my-sketch", sketch, 1 * u.m)
```

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```

>>> blocks = body.tessellate()
>>> blocks
>>> MultiBlock(0x7F94EC757460)
  N Blocks: 6
  X Bounds: 0.000, 4.000
  Y Bounds: -1.000, 0.000
  Z Bounds: -0.500, 4.500

```

Merge the body:

```

>>> mesh = body.tessellate(merge=True)
>>> mesh
PolyData (0x7f94ec75f3a0)
  N Cells: 12
  N Points: 24
  X Bounds: 0.000e+00, 4.000e+00
  Y Bounds: -1.000e+00, 0.000e+00
  Z Bounds: -5.000e-01, 4.500e+00
  N Arrays: 0

```

abstract `IBody.shell_body`(*offset: ansys.geometry.core.typing.Real*) → `bool`

Shell the body to the thickness specified.

Parameters

`offset`

[Real] Shell thickness.

Returns

`bool`

True when successful, False when failed.

abstract `IBody.remove_faces`(*selection: ansys.geometry.core.designer.face.Face | collections.abc.Iterable[ansys.geometry.core.designer.face.Face], offset: ansys.geometry.core.typing.Real*) → `bool`

Shell by removing a given set of faces.

Parameters

`selection`

[Face | Iterable[Face]] Face or faces to be removed.

`offset`

[Real] Shell thickness.

Returns

`bool`

True when successful, False when failed.

abstract `IBody.plot`(*merge: bool = True, screenshot: str | None = None, use_trame: bool | None = None, use_service_colors: bool | None = None, **plotting_options: dict | None*) → `None`

Plot the body.

Parameters

`merge`

[bool, default: True] Whether to merge the body into a single mesh. Performance improved

when True. When True (default), the individual faces of the tessellation are merged. When False, the number of triangles are preserved and only the topology is merged.

screenshot

[str, default: None] Path for saving a screenshot of the image that is being represented.

use_frame

[bool, default: None] Whether to enable the use of `frame`. The default is None, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.

use_service_colors

[bool, default: None] Whether to use the colors assigned to the body in the service. The default is None, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

**plotting_options

[dict, default: None] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

Examples

Extrude a box centered at the origin to create rectangular body and plot it:

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> mycomp = design.add_component("my-comp")
>>> body = mycomp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> body.plot()
```

Plot the body and color each face individually:

```
>>> body.plot(multi_colors=True)
```

`IBody.intersect(other: Body | collections.abc.Iterable[Body], keep_other: bool = False) → None`

Intersect two (or more) bodies.

Parameters

other

[Body] Body to intersect with.

keep_other

[bool, default: False] Whether to retain the intersected body or not.

Raises

ValueError

If the bodies do not intersect.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the intersected body is retained.

`IBody.subtract(other: Body | collections.abc.Iterable[Body], keep_other: bool = False) → None`

Subtract two (or more) bodies.

Parameters

`other`

[Body] Body to subtract from the `self` parameter.

`keep_other`

[bool, default: `False`] Whether to retain the subtracted body or not.

Raises

`ValueError`

If the subtraction results in an empty (complete) subtraction.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the subtracted body is retained.

`IBody.unite(other: Body | collections.abc.Iterable[Body], keep_other: bool = False) → None`

Unite two (or more) bodies.

Parameters

`other`

[Body] Body to unite with the `self` parameter.

`keep_other`

[bool, default: `False`] Whether to retain the united body or not.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the united body is retained.

MasterBody

```
class ansys.geometry.core.designer.body.MasterBody(id: str, name: str, grpc_client: an-
sys.geometry.core.connection.client.GrpcClient,
is_surface: bool = False)
```

Bases: `IBody`

Represents solids and surfaces organized within the design assembly.

Solids and surfaces synchronize to a design within a supporting Geometry service instance.

Parameters

`id`

[str] Server-defined ID for the body.

`name`

[str] User-defined label for the body.

parent_component

[Component] Parent component to place the new component under within the design assembly.

grpc_client

[GrpcClient] Active supporting geometry service instance for design modeling.

is_surface

[bool, default: False] Whether the master body is a surface or an 3D object (with volume). The default is False, in which case the master body is a surface. When True, the master body is a 3D object (with volume).

Overview**Abstract methods**

<i>imprint_curves</i>	Imprint all specified geometries onto specified faces of the body.
<i>project_curves</i>	Project all specified geometries onto the body.
<i>imprint_projected_curves</i>	Project and imprint specified geometries onto the body.
<i>plot</i>	Plot the body.
<i>intersect</i>	Intersect two (or more) bodies.
<i>subtract</i>	Subtract two (or more) bodies.
<i>unite</i>	Unite two (or more) bodies.

Methods

<i>reset_tessellation_cache</i>	Decorate MasterBody methods that need tessellation cache update.
<i>assign_material</i>	Assign a material against the active design.
<i>get_assigned_material</i>	Get the assigned material of the body.
<i>add_midsurface_thickness</i>	Add a mid-surface thickness to a surface body.
<i>add_midsurface_offset</i>	Add a mid-surface offset to a surface body.
<i>translate</i>	Translate the body in a specified direction and distance.
<i>set_name</i>	Set the name of the body.
<i>set_fill_style</i>	Set the fill style of the body.
<i>set_suppressed</i>	Set the body suppression state.
<i>set_color</i>	Set the color of the body.
<i>rotate</i>	Rotate the geometry body around the specified axis by a given angle.
<i>scale</i>	Scale the geometry body by the given value.
<i>map</i>	Map the geometry body to the new specified frame.
<i>mirror</i>	Mirror the geometry body across the specified plane.
<i>get_collision</i>	Get the collision state between bodies.
<i>copy</i>	Create a copy of the body under the specified parent.
<i>tessellate</i>	Tessellate the body and return the geometry as triangles.
<i>shell_body</i>	Shell the body to the thickness specified.
<i>remove_faces</i>	Shell by removing a given set of faces.

Properties

<code>id</code>	Get the ID of the body as a string.
<code>name</code>	Get the name of the body.
<code>fill_style</code>	Get the fill style of the body.
<code>is_suppressed</code>	Get the body suppression state.
<code>color</code>	Get the current color of the body.
<code>is_surface</code>	Check if the body is a planar body.
<code>surface_thickness</code>	Get the surface thickness of a surface body.
<code>surface_offset</code>	Get the surface offset type of a surface body.
<code>faces</code>	Get a list of all faces within the body.
<code>edges</code>	Get a list of all edges within the body.
<code>is_alive</code>	Check if the body is still alive and has not been deleted.
<code>volume</code>	Calculate the volume of the body.
<code>material</code>	Get the assigned material of the body.

Special methods

<code>__repr__</code>	Represent the master body as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.designer.body import MasterBody
```

Property detail

property `MasterBody.id`: `str`

Get the ID of the body as a string.

property `MasterBody.name`: `str`

Get the name of the body.

property `MasterBody.fill_style`: `str`

Get the fill style of the body.

property `MasterBody.is_suppressed`: `bool`

Get the body suppression state.

property `MasterBody.color`: `str`

Get the current color of the body.

property `MasterBody.is_surface`: `bool`

Check if the body is a planar body.

property `MasterBody.surface_thickness`: `pint.Quantity` | `None`

Get the surface thickness of a surface body.

Notes

This method is only for surface-type bodies that have been assigned a surface thickness.

property `MasterBody.surface_offset`: `MidSurfaceOffsetType` | `None`

Get the surface offset type of a surface body.

Notes

This method is only for surface-type bodies that have been assigned a surface offset.

property `MasterBody.faces`: `list[ansys.geometry.core.designer.face.Face]`

Get a list of all faces within the body.

Returns

`list[Face]`

property `MasterBody.edges`: `list[ansys.geometry.core.designer.edge.Edge]`

Get a list of all edges within the body.

Returns

`list[Edge]`

property `MasterBody.is_alive`: `bool`

Check if the body is still alive and has not been deleted.

property `MasterBody.volume`: `pint.Quantity`

Calculate the volume of the body.

Notes

When dealing with a planar surface, a value of 0 is returned as a volume.

property `MasterBody.material`: `ansys.geometry.core.materials.material.Material`

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

Method detail

`MasterBody.reset_tessellation_cache()`

Decorate `MasterBody` methods that need tessellation cache update.

Parameters

func

[method] Method to call.

Returns

Any

Output of the method, if any.

`MasterBody.assign_material(material: ansys.geometry.core.materials.material.Material) → None`

Assign a material against the active design.

Parameters

material

[Material] Source material data.

`MasterBody.get_assigned_material()` → `ansys.geometry.core.materials.material.Material`

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

`MasterBody.add_midsurface_thickness(thickness: pint.Quantity) → None`

Add a mid-surface thickness to a surface body.

Parameters

thickness

[Quantity] Thickness to assign.

Notes

Only surface bodies are eligible for mid-surface thickness assignment.

`MasterBody.add_midsurface_offset(offset: MidSurfaceOffsetType) → None`

Add a mid-surface offset to a surface body.

Parameters

offset_type

[MidSurfaceOffsetType] Surface offset to assign.

Notes

Only surface bodies are eligible for mid-surface offset assignment.

abstract `MasterBody.imprint_curves(faces: list[ansys.geometry.core.designer.face.Face], sketch: ansys.geometry.core.sketch.sketch.Sketch) → tuple[list[ansys.geometry.core.designer.edge.Edge], list[ansys.geometry.core.designer.face.Face]]`

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: list[Face]

list of faces to imprint the curves of the sketch onto.

sketch: Sketch

All curves to imprint on the faces.

Returns

`tuple[list[Edge], list[Face]]`

All impacted edges and faces from the imprint operation.

abstract `MasterBody.project_curves(direction: ansys.geometry.core.math.vector.UnitVector3D, sketch: ansys.geometry.core.sketch.sketch.Sketch, closest_face: bool, only_one_curve: bool = False) → list[ansys.geometry.core.designer.face.Face]`

Project all specified geometries onto the body.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

Whether to project only one curve of the entire sketch. When True, only one curve is projected.

Returns

`list[Face]`

All faces from the project curves operation.

Notes

The `only_one_curve` parameter allows you to optimize the server call because projecting curves is an expensive operation. This reduces the workload on the server side.

```
abstract MasterBody.imprint_projected_curves(direction:
    ansys.geometry.core.math.vector.UnitVector3D, sketch:
    ansys.geometry.core.sketch.sketch.Sketch, closest_face:
    bool, only_one_curve: bool = False) →
    list[ansys.geometry.core.designer.face.Face]
```

Project and imprint specified geometries onto the body.

This method combines the `project_curves()` and `imprint_curves()` method into one method. It has higher performance than calling them back-to-back when dealing with many curves. Because it is a specialized function, this method only returns the faces (and not the edges) from the imprint operation.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

Whether to project only one curve of the entire sketch. When True, only one curve is projected.

Returns

`list[Face]`

All imprinted faces from the operation.

Notes

The `only_one_curve` parameter allows you to optimize the server call because projecting curves is an expensive operation. This reduces the workload on the server side.

```
MasterBody.translate(direction: ansys.geometry.core.math.vector.UnitVector3D, distance: pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real) →
    None
```

Translate the body in a specified direction and distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Distance (magnitude) of the translation.

Returns**None****MasterBody.set_name**(*name: str*) → **None**

Set the name of the body.

MasterBody.set_fill_style(*fill_style: FillStyle*) → **None**

Set the fill style of the body.

MasterBody.set_suppressed(*suppressed: bool*) → **None**

Set the body suppression state.

MasterBody.set_color(*color: str | tuple[float, float, float]*) → **None**

Set the color of the body.

MasterBody.rotate(*axis_origin: ansys.geometry.core.math.point.Point3D, axis_direction: ansys.geometry.core.math.vector.UnitVector3D, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real*) → **None**

Rotate the geometry body around the specified axis by a given angle.

Parameters**axis_origin: Point3D**

Origin of the rotational axis.

axis_direction: UnitVector3D

The axis of rotation.

angle: ~pint.Quantity | Angle | Real

Angle (magnitude) of the rotation.

Returns**None****MasterBody.scale**(*value: ansys.geometry.core.typing.Real*) → **None**

Scale the geometry body by the given value.

Parameters**value: Real**

Value to scale the body by.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

MasterBody.map(*frame: ansys.geometry.core.math.frame.Frame*) → **None**

Map the geometry body to the new specified frame.

Parameters**frame: Frame**

Structure defining the orientation of the body.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

`MasterBody.mirror(plane: ansys.geometry.core.math.plane.Plane) → None`

Mirror the geometry body across the specified plane.

Parameters

plane: Plane

Represents the mirror.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

`MasterBody.get_collision(body: Body) → CollisionType`

Get the collision state between bodies.

Parameters

body: Body

Object that the collision state is checked with.

Returns

CollisionType

Enum that defines the collision state between bodies.

`MasterBody.copy(parent: ansys.geometry.core.designer.component.Component, name: str = None) → Body`

Create a copy of the body under the specified parent.

Parameters

parent: Component

Parent component to place the new body under within the design assembly.

name: str

Name to give the new body.

Returns

Body

Copy of the body.

`MasterBody.tessellate(merge: bool = False, transform: ansys.geometry.core.math.matrix.Matrix44 = IDENTITY_MATRIX44) → pyvista.PolyData | pyvista.MultiBlock`

Tessellate the body and return the geometry as triangles.

Parameters

merge

[bool, default: False] Whether to merge the body into a single mesh. When False (default), the number of triangles are preserved and only the topology is merged. When True, the individual faces of the tessellation are merged.

Returns

PolyData, MultiBlock

Merged `pyvista.PolyData` if `merge=True` or a composite dataset.

Examples

Extrude a box centered at the origin to create a rectangular body and tessellate it:

```

>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> my_comp = design.add_component("my-comp")
>>> body = my_comp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> blocks = body.tessellate()
>>> blocks
MultiBlock(0x7F94EC757460)
  N Blocks: 6
  X Bounds: 0.000, 4.000
  Y Bounds: -1.000, 0.000
  Z Bounds: -0.500, 4.500

```

Merge the body:

```

>>> mesh = body.tessellate(merge=True)
>>> mesh
PolyData (0x7f94ec75f3a0)
  N Cells: 12
  N Points: 24
  X Bounds: 0.000e+00, 4.000e+00
  Y Bounds: -1.000e+00, 0.000e+00
  Z Bounds: -5.000e-01, 4.500e+00
  N Arrays: 0

```

MasterBody.**shell_body**(*offset*: ansys.geometry.core.typing.Real) → bool

Shell the body to the thickness specified.

Parameters

offset

[Real] Shell thickness.

Returns

bool

True when successful, False when failed.

MasterBody.**remove_faces**(*selection*: ansys.geometry.core.designer.face.Face |
collections.abc.Iterable[ansys.geometry.core.designer.face.Face], *offset*:
ansys.geometry.core.typing.Real) → bool

Shell by removing a given set of faces.

Parameters

selection

[Face | Iterable[Face]] Face or faces to be removed.

offset

[Real] Shell thickness.

Returns**bool**

True when successful, False when failed.

abstract `MasterBody.plot`(*merge*: `bool = True`, *screenshot*: `str | None = None`, *use_frame*: `bool | None = None`, *use_service_colors*: `bool | None = None`, ***plotting_options*: `dict | None`) → `None`

Plot the body.

Parameters**merge**

[`bool`, default: `True`] Whether to merge the body into a single mesh. Performance improved when `True`. When `True` (default), the individual faces of the tessellation are merged. When `False`, the number of triangles are preserved and only the topology is merged.

screenshot

[`str`, default: `None`] Path for saving a screenshot of the image that is being represented.

use_frame

[`bool`, default: `None`] Whether to enable the use of `frame`. The default is `None`, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.

use_service_colors

[`bool`, default: `None`] Whether to use the colors assigned to the body in the service. The default is `None`, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

****plotting_options**

[`dict`, default: `None`] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

Examples

Extrude a box centered at the origin to create rectangular body and plot it:

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> mycomp = design.add_component("my-comp")
>>> body = mycomp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> body.plot()
```

Plot the body and color each face individually:

```
>>> body.plot(multi_colors=True)
```

abstract `MasterBody.intersect`(*other*: `Body | collections.abc.Iterable[Body]`, *keep_other*: `bool = False`) → `None`

Intersect two (or more) bodies.

Parameters

other

[Body] Body to intersect with.

keep_other

[bool, default: `False`] Whether to retain the intersected body or not.

Raises**ValueError**

If the bodies do not intersect.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the intersected body is retained.

abstract `MasterBody.subtract`(*other*: Body | *collections.abc.Iterable*[Body], *keep_other*: bool = *False*) → None

Subtract two (or more) bodies.

Parameters**other**

[Body] Body to subtract from the `self` parameter.

keep_other

[bool, default: `False`] Whether to retain the subtracted body or not.

Raises**ValueError**

If the subtraction results in an empty (complete) subtraction.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the subtracted body is retained.

abstract `MasterBody.unite`(*other*: Body | *collections.abc.Iterable*[Body], *keep_other*: bool = *False*) → None

Unite two (or more) bodies.

Parameters**other**

[Body] Body to unite with the `self` parameter.

keep_other

[bool, default: `False`] Whether to retain the united body or not.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the united body is retained.

`MasterBody.__repr__`() → str

Represent the master body as a string.

Body

class `ansys.geometry.core.designer.body.Body`(*id*, *name*, *parent_component*:
`ansys.geometry.core.designer.component.Component`,
template: `MasterBody`)

Bases: `IBody`

Represents solids and surfaces organized within the design assembly.

Solids and surfaces synchronize to a design within a supporting Geometry service instance.

Parameters

id

[`str`] Server-defined ID for the body.

name

[`str`] User-defined label for the body.

parent_component

[`Component`] Parent component to place the new component under within the design assembly.

template

[`MasterBody`] Master body that this body is an occurrence of.

Overview

Methods

<code>reset_tessellation_cache</code>	Decorate Body methods that require a tessellation cache update.
<code>assign_material</code>	Assign a material against the active design.
<code>get_assigned_material</code>	Get the assigned material of the body.
<code>add_midsurface_thickness</code>	Add a mid-surface thickness to a surface body.
<code>add_midsurface_offset</code>	Add a mid-surface offset to a surface body.
<code>imprint_curves</code>	Imprint all specified geometries onto specified faces of the body.
<code>project_curves</code>	Project all specified geometries onto the body.
<code>imprint_projected_curves</code>	Project and imprint specified geometries onto the body.
<code>set_name</code>	Set the name of the body.
<code>set_fill_style</code>	Set the fill style of the body.
<code>set_suppressed</code>	Set the body suppression state.
<code>set_color</code>	Set the color of the body.
<code>translate</code>	Translate the body in a specified direction and distance.
<code>rotate</code>	Rotate the geometry body around the specified axis by a given angle.
<code>scale</code>	Scale the geometry body by the given value.
<code>map</code>	Map the geometry body to the new specified frame.
<code>mirror</code>	Mirror the geometry body across the specified plane.
<code>get_collision</code>	Get the collision state between bodies.
<code>copy</code>	Create a copy of the body under the specified parent.
<code>tessellate</code>	Tessellate the body and return the geometry as triangles.
<code>shell_body</code>	Shell the body to the thickness specified.
<code>remove_faces</code>	Shell by removing a given set of faces.
<code>plot</code>	Plot the body.
<code>intersect</code>	Intersect two (or more) bodies.
<code>subtract</code>	Subtract two (or more) bodies.
<code>unite</code>	Unite two (or more) bodies.

Properties

<code>id</code>	Get the ID of the body as a string.
<code>name</code>	Get the name of the body.
<code>fill_style</code>	Get the fill style of the body.
<code>is_suppressed</code>	Get the body suppression state.
<code>color</code>	Get the color of the body.
<code>parent_component</code>	
<code>faces</code>	Get a list of all faces within the body.
<code>edges</code>	Get a list of all edges within the body.
<code>is_alive</code>	Check if the body is still alive and has not been deleted.
<code>is_surface</code>	Check if the body is a planar body.
<code>surface_thickness</code>	Get the surface thickness of a surface body.
<code>surface_offset</code>	Get the surface offset type of a surface body.
<code>volume</code>	Calculate the volume of the body.
<code>material</code>	Get the assigned material of the body.

Special methods

<code>__repr__</code>	Represent the Body as a string.
-----------------------	---------------------------------

Import detail

```
from ansys.geometry.core.designer.body import Body
```

Property detail

property `Body.id: str`

Get the ID of the body as a string.

property `Body.name: str`

Get the name of the body.

property `Body.fill_style: str`

Get the fill style of the body.

property `Body.is_suppressed: bool`

Get the body suppression state.

property `Body.color: str`

Get the color of the body.

property `Body.parent_component: ansys.geometry.core.designer.component.Component`

property `Body.faces: list[ansys.geometry.core.designer.face.Face]`

Get a list of all faces within the body.

Returns

`list[Face]`

property `Body.edges: list[ansys.geometry.core.designer.edge.Edge]`

Get a list of all edges within the body.

Returns`list[Edge]`**property** `Body.is_alive: bool`

Check if the body is still alive and has not been deleted.

property `Body.is_surface: bool`

Check if the body is a planar body.

property `Body.surface_thickness: pint.Quantity | None`

Get the surface thickness of a surface body.

Notes

This method is only for surface-type bodies that have been assigned a surface thickness.

property `Body.surface_offset: MidSurfaceOffsetType | None`

Get the surface offset type of a surface body.

Notes

This method is only for surface-type bodies that have been assigned a surface offset.

property `Body.volume: pint.Quantity`

Calculate the volume of the body.

Notes

When dealing with a planar surface, a value of 0 is returned as a volume.

property `Body.material: ansys.geometry.core.materials.material.Material`

Get the assigned material of the body.

Returns**Material**

Material assigned to the body.

Method detail`Body.reset_tessellation_cache()`

Decorate Body methods that require a tessellation cache update.

Parameters**func**

[method] Method to call.

Returns**Any**

Output of the method, if any.

`Body.assign_material(material: ansys.geometry.core.materials.material.Material) → None`

Assign a material against the active design.

Parameters**material**

[Material] Source material data.

Body.**get_assigned_material**() → *ansys.geometry.core.materials.material.Material*

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

Body.**add_midsurface_thickness**(*thickness: pint.Quantity*) → *None*

Add a mid-surface thickness to a surface body.

Parameters

thickness

[*Quantity*] Thickness to assign.

Notes

Only surface bodies are eligible for mid-surface thickness assignment.

Body.**add_midsurface_offset**(*offset: MidSurfaceOffsetType*) → *None*

Add a mid-surface offset to a surface body.

Parameters

offset_type

[*MidSurfaceOffsetType*] Surface offset to assign.

Notes

Only surface bodies are eligible for mid-surface offset assignment.

Body.**imprint_curves**(*faces: list[ansys.geometry.core.designer.face.Face]*, *sketch: ansys.geometry.core.sketch.sketch.Sketch*) → *tuple[list[ansys.geometry.core.designer.edge.Edge], list[ansys.geometry.core.designer.face.Face]]*

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: list[Face]

list of faces to imprint the curves of the sketch onto.

sketch: Sketch

All curves to imprint on the faces.

Returns

tuple[list[Edge], list[Face]]

All impacted edges and faces from the imprint operation.

Body.**project_curves**(*direction: ansys.geometry.core.math.vector.UnitVector3D*, *sketch: ansys.geometry.core.sketch.sketch.Sketch*, *closest_face: bool*, *only_one_curve: bool = False*) → *list[ansys.geometry.core.designer.face.Face]*

Project all specified geometries onto the body.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

Whether to project only one curve of the entire sketch. When True, only one curve is projected.

Returns

list[Face]

All faces from the project curves operation.

Notes

The `only_one_curve` parameter allows you to optimize the server call because projecting curves is an expensive operation. This reduces the workload on the server side.

Body.**imprint_projected_curves**(*direction: ansys.geometry.core.math.vector.UnitVector3D, sketch: ansys.geometry.core.sketch.sketch.Sketch, closest_face: bool, only_one_curve: bool = False*) → *list[ansys.geometry.core.designer.face.Face]*

Project and imprint specified geometries onto the body.

This method combines the `project_curves()` and `imprint_curves()` method into one method. It has higher performance than calling them back-to-back when dealing with many curves. Because it is a specialized function, this method only returns the faces (and not the edges) from the imprint operation.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

Whether to project only one curve of the entire sketch. When True, only one curve is projected.

Returns

list[Face]

All imprinted faces from the operation.

Notes

The `only_one_curve` parameter allows you to optimize the server call because projecting curves is an expensive operation. This reduces the workload on the server side.

Body.**set_name**(*name: str*) → *None*

Set the name of the body.

Body.**set_fill_style**(*fill_style: FillStyle*) → *None*

Set the fill style of the body.

Body.**set_suppressed**(*suppressed: bool*) → None

Set the body suppression state.

Body.**set_color**(*color: str | tuple[float, float, float]*) → None

Set the color of the body.

Parameters

color

[*str* | *tuple*[*float*, *float*, *float*]] Color to set the body to. This can be a string representing a color name or a tuple of RGB values in the range [0, 1] (RGBA) or [0, 255] (pure RGB).

Body.**translate**(*direction: ansys.geometry.core.math.vector.UnitVector3D*, *distance: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real*) → None

Translate the body in a specified direction and distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Distance (magnitude) of the translation.

Returns

None

Body.**rotate**(*axis_origin: ansys.geometry.core.math.point.Point3D*, *axis_direction: ansys.geometry.core.math.vector.UnitVector3D*, *angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real*) → None

Rotate the geometry body around the specified axis by a given angle.

Parameters

axis_origin: Point3D

Origin of the rotational axis.

axis_direction: UnitVector3D

The axis of rotation.

angle: ~pint.Quantity | Angle | Real

Angle (magnitude) of the rotation.

Returns

None

Body.**scale**(*value: ansys.geometry.core.typing.Real*) → None

Scale the geometry body by the given value.

Parameters

value: Real

Value to scale the body by.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

Body.**map**(*frame*: ansys.geometry.core.math.frame.Frame) → None

Map the geometry body to the new specified frame.

Parameters

frame: Frame

Structure defining the orientation of the body.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

Body.**mirror**(*plane*: ansys.geometry.core.math.plane.Plane) → None

Mirror the geometry body across the specified plane.

Parameters

plane: Plane

Represents the mirror.

Notes

The calling object is directly modified when this method is called. Thus, it is important to make copies if needed.

Body.**get_collision**(*body*: Body) → *CollisionType*

Get the collision state between bodies.

Parameters

body: Body

Object that the collision state is checked with.

Returns

CollisionType

Enum that defines the collision state between bodies.

Body.**copy**(*parent*: ansys.geometry.core.designer.component.Component, *name*: *str* = None) → Body

Create a copy of the body under the specified parent.

Parameters

parent: Component

Parent component to place the new body under within the design assembly.

name: str

Name to give the new body.

Returns

Body

Copy of the body.

Body.**tessellate**(*merge*: *bool* = False) → *pyvista.PolyData* | *pyvista.MultiBlock*

Tessellate the body and return the geometry as triangles.

Parameters

merge

[*bool*, default: *False*] Whether to merge the body into a single mesh. When *False* (default), the number of triangles are preserved and only the topology is merged. When *True*, the individual faces of the tessellation are merged.

Returns**PolyData, MultiBlock**Merged `pyvista.PolyData` if `merge=True` or a composite dataset.**Examples**

Extrude a box centered at the origin to create a rectangular body and tessellate it:

```

>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> my_comp = design.add_component("my-comp")
>>> body = my_comp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> blocks = body.tessellate()
>>> blocks
>>> MultiBlock(0x7F94EC757460)
  N Blocks: 6
  X Bounds: 0.000, 4.000
  Y Bounds: -1.000, 0.000
  Z Bounds: -0.500, 4.500

```

Merge the body:

```

>>> mesh = body.tessellate(merge=True)
>>> mesh
PolyData (0x7f94ec75f3a0)
  N Cells: 12
  N Points: 24
  X Bounds: 0.000e+00, 4.000e+00
  Y Bounds: -1.000e+00, 0.000e+00
  Z Bounds: -5.000e-01, 4.500e+00
  N Arrays: 0

```

Body.**shell_body**(*offset*: *ansys.geometry.core.typing.Real*) → *bool*

Shell the body to the thickness specified.

Parameters**offset**

[Real] Shell thickness.

Returns**bool**

True when successful, False when failed.

Body.**remove_faces**(*selection*: *ansys.geometry.core.designer.face.Face* | *collections.abc.Iterable[ansys.geometry.core.designer.face.Face]*, *offset*: *ansys.geometry.core.typing.Real*) → *bool*

Shell by removing a given set of faces.

Parameters

selection

[Face | Iterable[Face]] Face or faces to be removed.

offset

[Real] Shell thickness.

Returns

bool

True when successful, False when failed.

`Body.plot(merge: bool = True, screenshot: str | None = None, use_frame: bool | None = None, use_service_colors: bool | None = None, **plotting_options: dict | None) → None`

Plot the body.

Parameters

merge

[bool, default: True] Whether to merge the body into a single mesh. Performance improved when True. When True (default), the individual faces of the tessellation are merged. When False, the number of triangles are preserved and only the topology is merged.

screenshot

[str, default: None] Path for saving a screenshot of the image that is being represented.

use_frame

[bool, default: None] Whether to enable the use of `frame`. The default is None, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.

use_service_colors

[bool, default: None] Whether to use the colors assigned to the body in the service. The default is None, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

**plotting_options

[dict, default: None] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

Examples

Extrude a box centered at the origin to create rectangular body and plot it:

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> mycomp = design.add_component("my-comp")
>>> body = mycomp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> body.plot()
```

Plot the body and color each face individually:

```
>>> body.plot(multi_colors=True)
```

Body.**intersect**(*other*: Body | *collections.abc.Iterable*[Body], *keep_other*: bool = False) → None

Intersect two (or more) bodies.

Parameters

other

[Body] Body to intersect with.

keep_other

[bool, default: False] Whether to retain the intersected body or not.

Raises

ValueError

If the bodies do not intersect.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the intersected body is retained.

Body.**subtract**(*other*: Body | *collections.abc.Iterable*[Body], *keep_other*: bool = False) → None

Subtract two (or more) bodies.

Parameters

other

[Body] Body to subtract from the `self` parameter.

keep_other

[bool, default: False] Whether to retain the subtracted body or not.

Raises

ValueError

If the subtraction results in an empty (complete) subtraction.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the subtracted body is retained.

Body.**unite**(*other*: Body | *collections.abc.Iterable*[Body], *keep_other*: bool = False) → None

Unite two (or more) bodies.

Parameters

other

[Body] Body to unite with the `self` parameter.

keep_other

[bool, default: False] Whether to retain the united body or not.

Notes

The `self` parameter is directly modified with the result, and the `other` parameter is consumed. Thus, it is important to make copies if needed. If the `keep_other` parameter is set to `True`, the united body is retained.

`Body.__repr__()` → `str`

Represent the `Body` as a string.

MidSurfaceOffsetType

`class ansys.geometry.core.designer.body.MidSurfaceOffsetType(*args, **kws)`

Bases: `enum.Enum`

Provides values for mid-surface offsets supported.

Overview

Attributes

<code>MIDDLE</code>
<code>TOP</code>
<code>BOTTOM</code>
<code>VARIABLE</code>
<code>CUSTOM</code>

Import detail

```
from ansys.geometry.core.designer.body import MidSurfaceOffsetType
```

Attribute detail

`MidSurfaceOffsetType.MIDDLE = 0`

`MidSurfaceOffsetType.TOP = 1`

`MidSurfaceOffsetType.BOTTOM = 2`

`MidSurfaceOffsetType.VARIABLE = 3`

`MidSurfaceOffsetType.CUSTOM = 4`

CollisionType

`class ansys.geometry.core.designer.body.CollisionType(*args, **kws)`

Bases: `enum.Enum`

Provides values for collision types between bodies.

Overview

Attributes

<i>NONE</i>
<i>TOUCH</i>
<i>INTERSECT</i>
<i>CONTAINED</i>
<i>CONTAINEDTOUCH</i>

Import detail

```
from ansys.geometry.core.designer.body import CollisionType
```

Attribute detail

CollisionType.**NONE** = 0

CollisionType.**TOUCH** = 1

CollisionType.**INTERSECT** = 2

CollisionType.**CONTAINED** = 3

CollisionType.**CONTAINEDTOUCH** = 4

FillStyle

```
class ansys.geometry.core.designer.body.FillStyle(*args, **kwargs)
```

Bases: `enum.Enum`

Provides values for fill styles supported.

Overview

Attributes

<i>DEFAULT</i>
<i>OPAQUE</i>
<i>TRANSPARENT</i>

Import detail

```
from ansys.geometry.core.designer.body import FillStyle
```

Attribute detail

FillStyle.**DEFAULT** = 0

FillStyle.**OPAQUE** = 1

FillStyle.**TRANSPARENT** = 2

Description

Provides for managing a body.

Module detail

body.__TEMPORARY_BOOL_OPS_FIX__ = (99, 0, 0)

The component.py module

Summary

Classes

<i>Component</i>	Provides for creating and managing a component.
------------------	---

Enums

<i>SharedTopologyType</i>	Shared topologies available.
<i>ExtrusionDirection</i>	Enum for extrusion direction definition.

Component

```
class ansys.geometry.core.designer.component.Component(name: str, parent_component: Component | None, grpc_client: ansys.geometry.core.connection.client.GrpcClient, template: Component | None = None, instance_name: str | None = None, preexisting_id: str | None = None, master_component: ansys.geometry.core.designer.part.MasterComponent | None = None, read_existing_comp: bool = False)
```

Provides for creating and managing a component.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

name

[str] User-defined label for the new component.

parent_component

[Component or None] Parent component to place the new component under within the design assembly. The default is None only when dealing with a Design object.

grpc_client

[GrpcClient] Active supporting Geometry service instance for design modeling.

template

[Component, default: None] Template to create this component from. This creates an instance component that shares a master with the template component.

instance_name: str, default: None

User defined optional name for the component instance.

preexisting_id

[*str*, default: *None*] ID of a component pre-existing on the server side to use to create the component on the client-side data model. If an ID is specified, a new component is not created on the server.

master_component

[*MasterComponent*, default: *None*] Master component to use to create a nested component instance instead of creating a new component.

read_existing_comp

[*bool*, default: *False*] Whether an existing component on the service should be read. This parameter is only valid when connecting to an existing service session. Otherwise, avoid using this optional parameter.

Overview

Methods

<i>get_all_bodies</i>	Get all bodies in the component hierarchy.
<i>get_world_transform</i>	Get the full transformation matrix of the component in world space.
<i>modify_placement</i>	Apply a translation and/or rotation to the placement matrix.
<i>reset_placement</i>	Reset a component's placement matrix to an identity matrix.
<i>add_component</i>	Add a new component under this component within the design assembly.
<i>set_shared_topology</i>	Set the shared topology to apply to the component.
<i>extrude_sketch</i>	Create a solid body by extruding the sketch profile a distance.
<i>sweep_sketch</i>	Create a body by sweeping a planar profile along a path.
<i>sweep_chain</i>	Create a body by sweeping a chain of curves along a path.
<i>revolve_sketch</i>	Create a solid body by revolving a sketch profile around an axis.
<i>extrude_face</i>	Extrude the face profile by a given distance to create a solid body.
<i>create_sphere</i>	Create a sphere body defined by the center point and the radius.
<i>create_body_from_loft_profile</i>	Create a lofted body from a collection of trimmed curves.
<i>create_surface</i>	Create a surface body with a sketch profile.
<i>create_surface_from_face</i>	Create a surface body based on a face.
<i>create_body_from_surface</i>	Create a surface body from a trimmed surface.
<i>create_surface_from_trimmed_curves</i>	Create a surface body from a list of trimmed curves all lying on the same plane.
<i>create_coordinate_system</i>	Create a coordinate system.
<i>translate_bodies</i>	Translate the bodies in a specified direction by a distance.
<i>create_beams</i>	Create beams under the component.
<i>create_beam</i>	Create a beam under the component.
<i>delete_component</i>	Delete a component (itself or its children).
<i>delete_body</i>	Delete a body belonging to this component (or its children).
<i>add_design_point</i>	Create a single design point.
<i>add_design_points</i>	Create a list of design points.
<i>delete_beam</i>	Delete an existing beam belonging to this component's scope.
<i>search_component</i>	Search nested components recursively for a component.
<i>search_body</i>	Search bodies in the component's scope.
<i>search_beam</i>	Search beams in the component's scope.
<i>tessellate</i>	Tessellate the component.
<i>plot</i>	Plot the component.
<i>tree_print</i>	Print the component in tree format.

Properties

<i>id</i>	ID of the component.
<i>name</i>	Name of the component.
<i>instance_name</i>	Name of the component instance.
<i>components</i>	List of Component objects inside of the component.
<i>bodies</i>	List of Body objects inside of the component.
<i>beams</i>	List of Beam objects inside of the component.
<i>design_points</i>	List of DesignPoint objects inside of the component.
<i>coordinate_systems</i>	List of CoordinateSystem objects inside of the component.
<i>parent_component</i>	Parent of the component.
<i>is_alive</i>	Whether the component is still alive on the server side.
<i>shared_topology</i>	Shared topology type of the component (if any).

Special methods

<code>__repr__</code>	Represent the Component as a string.
-----------------------	--------------------------------------

Import detail

```
from ansys.geometry.core.designer.component import Component
```

Property detail

property Component.**id**: **str**

ID of the component.

property Component.**name**: **str**

Name of the component.

property Component.**instance_name**: **str**

Name of the component instance.

property Component.**components**: **list**[Component]

List of Component objects inside of the component.

property Component.**bodies**: **list**[ansys.geometry.core.designer.body.Body]

List of Body objects inside of the component.

property Component.**beams**: **list**[ansys.geometry.core.designer.beam.Beam]

List of Beam objects inside of the component.

property Component.**design_points**:

list[ansys.geometry.core.designer.designpoint.DesignPoint]

List of DesignPoint objects inside of the component.

property Component.**coordinate_systems**:

list[ansys.geometry.core.designer.coordinate_system.CoordinateSystem]

List of CoordinateSystem objects inside of the component.

property Component.**parent_component**: Component

Parent of the component.

property `Component.is_alive`: **bool**

Whether the component is still alive on the server side.

property `Component.shared_topology`: **SharedTopologyType | None**

Shared topology type of the component (if any).

Notes

If no shared topology has been set, `None` is returned.

Method detail

`Component.get_all_bodies()` → `list[ansys.geometry.core.designer.body.Body]`

Get all bodies in the component hierarchy.

Returns

`list[Body]`

List of all bodies in the component hierarchy.

`Component.get_world_transform()` → `ansys.geometry.core.math.matrix.Matrix44`

Get the full transformation matrix of the component in world space.

Returns

Matrix44

4x4 transformation matrix of the component in world space.

`Component.modify_placement`(*translation*: `ansys.geometry.core.math.vector.Vector3D | None = None`,
rotation_origin: `ansys.geometry.core.math.point.Point3D | None = None`,
rotation_direction: `ansys.geometry.core.math.vector.UnitVector3D | None = None`,
rotation_angle: `pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0`)

Apply a translation and/or rotation to the placement matrix.

Parameters

translation

[`Vector3D`, default: `None`] Vector that defines the desired translation to the component.

rotation_origin

[`Point3D`, default: `None`] Origin that defines the axis to rotate the component about.

rotation_direction

[`UnitVector3D`, default: `None`] Direction of the axis to rotate the component about.

rotation_angle

[`Quantity | Angle | Real`, default: `0`] Angle to rotate the component around the axis.

Notes

To reset a component's placement to an identity matrix, see `reset_placement()` or call `modify_placement()` with no arguments.

`Component.reset_placement()`

Reset a component's placement matrix to an identity matrix.

See `modify_placement()`.

`Component.add_component`(*name*: *str*, *template*: Component | *None* = *None*, *instance_name*: *str* = *None*) → *Component*

Add a new component under this component within the design assembly.

Parameters

name

[*str*] User-defined label for the new component.

template

[Component, default: *None*] Template to create this component from. This creates an instance component that shares a master with the template component.

Returns

Component

New component with no children in the design assembly.

`Component.set_shared_topology`(*share_type*: SharedTopologyType) → *None*

Set the shared topology to apply to the component.

Parameters

share_type

[SharedTopologyType] Shared topology type to assign to the component.

`Component.extrude_sketch`(*name*: *str*, *sketch*: ansys.geometry.core.sketch.sketch.Sketch, *distance*: *pint.Quantity* | ansys.geometry.core.misc.measurements.Distance | *ansys.geometry.core.typing.Real*, *direction*: ExtrusionDirection | *str* = *ExtrusionDirection.POSITIVE*, *cut*: *bool* = *False*) → *ansys.geometry.core.designer.body.Body* | *None*

Create a solid body by extruding the sketch profile a distance.

Parameters

name

[*str*] User-defined label for the new solid body.

sketch

[Sketch] Two-dimensional sketch source for the extrusion.

distance

[*Quantity* | *Distance* | *Real*] Distance to extrude the solid body.

direction

[ExtrusionDirection | *str*, default: "+"] Direction for extruding the solid body. The default is to extrude in the positive normal direction of the sketch. Options are "+" and "-" as a string, or the enum values.

cut

[*bool*, default: *False*] Whether to cut the extrusion from the existing component. By default, the extrusion is added to the existing component.

Returns

Body

Extruded body from the given sketch.

None

If the cut parameter is True, the function returns None.

Notes

The newly created body is placed under this component within the design assembly.

Component `.sweep_sketch` (*name*: *str*, *sketch*: `ansys.geometry.core.sketch.sketch.Sketch`, *path*: `list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]`) → `ansys.geometry.core.designer.body.Body`

Create a body by sweeping a planar profile along a path.

Parameters

name

[`str`] User-defined label for the new solid body.

sketch

[`Sketch`] Two-dimensional sketch source for the extrusion.

path

[`list[TrimmedCurve]`] The path to sweep the profile along.

Returns

Body

Created body from the given sketch.

Notes

The newly created body is placed under this component within the design assembly.

Component `.sweep_chain` (*name*: *str*, *path*: `list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]`, *chain*: `list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]`) → `ansys.geometry.core.designer.body.Body`

Create a body by sweeping a chain of curves along a path.

Parameters

name

[`str`] User-defined label for the new solid body.

path

[`list[TrimmedCurve]`] The path to sweep the chain along.

chain

[`list[TrimmedCurve]`] A chain of trimmed curves.

Returns

Body

Created body from the given sketch.

Notes

The newly created body is placed under this component within the design assembly.

Component `.revolve_sketch` (*name*: *str*, *sketch*: `ansys.geometry.core.sketch.sketch.Sketch`, *axis*: `ansys.geometry.core.math.vector.Vector3D`, *angle*: `pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real`, *rotation_origin*: `ansys.geometry.core.math.point.Point3D`) → `ansys.geometry.core.designer.body.Body`

Create a solid body by revolving a sketch profile around an axis.

Parameters**name**

[`str`] User-defined label for the new solid body.

sketch

[`Sketch`] Two-dimensional sketch source for the revolve.

axis

[`Vector3D`] Axis of rotation for the revolve.

angle

[`Quantity` | `Angle` | `Real`] Angle to revolve the solid body around the axis. The angle can be positive or negative.

rotation_origin

[`Point3D`] Origin of the axis of rotation.

Returns**Body**

Revolved body from the given sketch.

Component.`extrude_face`(*name*: `str`, *face*: `ansys.geometry.core.designer.face.Face`, *distance*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance`, *direction*: `ExtrusionDirection` | `str` = `ExtrusionDirection.POSITIVE`) → `ansys.geometry.core.designer.body.Body`

Extrude the face profile by a given distance to create a solid body.

There are no modifications against the body containing the source face.

Parameters**name**

[`str`] User-defined label for the new solid body.

face

[`Face`] Target face to use as the source for the new surface.

distance

[`Quantity` | `Distance` | `Real`] Distance to extrude the solid body.

direction

[`ExtrusionDirection` | `str`, default: “+”] Direction for extruding the solid body’s face. The default is to extrude in the positive normal direction of the face. Options are “+” and “-” as a string, or the enum values.

Returns**Body**

Extruded solid body.

Notes

The source face can be anywhere within the design component hierarchy. Therefore, there is no validation requiring that the face is placed under the target component where the body is to be created.

Component.`create_sphere`(*name*: `str`, *center*: `ansys.geometry.core.math.point.Point3D`, *radius*: `ansys.geometry.core.misc.measurements.Distance`) → `ansys.geometry.core.designer.body.Body`

Create a sphere body defined by the center point and the radius.

Parameters

name
[str] Body name.

center
[Point3D] Center point of the sphere.

radius
[Distance] Radius of the sphere.

Returns

Body
Sphere body object.

Component.**create_body_from_loft_profile**(*name: str, profiles: list[list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]], periodic: bool = False, ruled: bool = False*) → *ansys.geometry.core.designer.body.Body*

Create a lofted body from a collection of trimmed curves.

Parameters

name
[str] Name of the lofted body.

profiles
[list[list[TrimmedCurve]]] Collection of lists of trimmed curves (profiles) defining the lofted body's shape.

periodic
[bool, default: False] Whether the lofted body should have periodic continuity.

ruled
[bool] Whether the lofted body should be ruled.

Returns

Body
Created lofted body object.

Notes

Surfaces produced have a U parameter in the direction of the profile curves, and a V parameter in the direction of lofting. Profiles can have different numbers of segments. A minimum twist solution is produced. Profiles should be all closed or all open. Closed profiles cannot contain inner loops. If closed profiles are supplied, a closed (solid) body is produced, if possible. Otherwise, an open (sheet) body is produced. The periodic argument applies when the profiles are closed. It is ignored if the profiles are open.

If `periodic=True`, at least three profiles must be supplied. The loft continues from the last profile back to the first profile to produce surfaces that are periodic in V.

If `periodic=False`, at least two profiles must be supplied. If the first and last profiles are planar, end capping faces are created. Otherwise, an open (sheet) body is produced. If `ruled=True`, separate ruled surfaces are produced between each pair of profiles. If `periodic=True`, the loft continues from the last profile back to the first profile, but the surfaces are not periodic.

Component.**create_surface**(*name: str, sketch: ansys.geometry.core.sketch.sketch.Sketch*) → *ansys.geometry.core.designer.body.Body*

Create a surface body with a sketch profile.

The newly created body is placed under this component within the design assembly.

Parameters**name**

[*str*] User-defined label for the new surface body.

sketch

[Sketch] Two-dimensional sketch source for the surface definition.

Returns**Body**

Body (as a planar surface) from the given sketch.

Component.**create_surface_from_face**(*name: str, face: ansys.geometry.core.designer.face.Face*) →
ansys.geometry.core.designer.body.Body

Create a surface body based on a face.

Parameters**name**

[*str*] User-defined label for the new surface body.

face

[Face] Target face to use as the source for the new surface.

Returns**Body**

Surface body.

Notes

The source face can be anywhere within the design component hierarchy. Therefore, there is no validation requiring that the face is placed under the target component where the body is to be created.

Component.**create_body_from_surface**(*name: str, trimmed_surface: ansys.geometry.core.shapes_surfaces.TrimmedSurface*) →
ansys.geometry.core.designer.body.Body

Create a surface body from a trimmed surface.

Parameters**name**

[*str*] User-defined label for the new surface body.

trimmed_surface

[TrimmedSurface] Geometry for the new surface body.

Returns**Body**

Surface body.

Notes

It is possible to create a closed solid body (as opposed to an open surface body) with a Sphere or Torus if they are untrimmed. This can be validated with *body.is_surface*.

Component.**create_surface_from_trimmed_curves**(*name: str, trimmed_curves: list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]*) →
ansys.geometry.core.designer.body.Body

Create a surface body from a list of trimmed curves all lying on the same plane.

Parameters**name**

[[str](#)] User-defined label for the new surface body.

trimmed_curves

[[list](#)][[TrimmedCurve](#)] Curves to define the plane and body.

Returns**Body**

Surface body.

`Component.create_coordinate_system(name: str, frame: ansys.geometry.core.math.frame.Frame) → ansys.geometry.core.designer.coordinate_system.CoordinateSystem`

Create a coordinate system.

The newly created coordinate system is place under this component within the design assembly.

Parameters**name**

[[str](#)] User-defined label for the new coordinate system.

frame

[[Frame](#)] Frame defining the coordinate system bounds.

Returns**CoordinateSystem**

`Component.translate_bodies(bodies: list[ansys.geometry.core.designer.body.Body], direction: ansys.geometry.core.math.vector.UnitVector3D, distance: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real) → None`

Translate the bodies in a specified direction by a distance.

Parameters**bodies: list[Body]**

list of bodies to translate by the same distance.

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Magnitude of the translation.

Returns

[None](#)

Notes

If the body does not belong to this component (or its children), it is not translated.

`Component.create_beams(segments: list[tuple[ansys.geometry.core.math.point.Point3D, ansys.geometry.core.math.point.Point3D]], profile: ansys.geometry.core.designer.beam.BeamProfile) → list[ansys.geometry.core.designer.beam.Beam]`

Create beams under the component.

Parameters

segments

[list[tuple[Point3D, Point3D]]] list of start and end pairs, each specifying a single line segment.

profile

[BeamProfile] Beam profile to use to create the beams.

Notes

The newly created beams synchronize to a design within a supporting Geometry service instance.

Component.**create_beam**(*start*: ansys.geometry.core.math.point.Point3D, *end*: ansys.geometry.core.math.point.Point3D, *profile*: ansys.geometry.core.designer.beam.BeamProfile) → *ansys.geometry.core.designer.beam.Beam*

Create a beam under the component.

The newly created beam synchronizes to a design within a supporting Geometry service instance.

Parameters**start**

[Point3D] Starting point of the beam line segment.

end

[Point3D] Ending point of the beam line segment.

profile

[BeamProfile] Beam profile to use to create the beam.

Component.**delete_component**(*component*: Component | *str*) → None

Delete a component (itself or its children).

Parameters**component**

[Component | *str*] ID of the component or instance to delete.

Notes

If the component is not this component (or its children), it is not deleted.

Component.**delete_body**(*body*: ansys.geometry.core.designer.body.Body | *str*) → None

Delete a body belonging to this component (or its children).

Parameters**body**

[Body | *str*] ID of the body or instance to delete.

Notes

If the body does not belong to this component (or its children), it is not deleted.

Component.**add_design_point**(*name*: *str*, *point*: ansys.geometry.core.math.point.Point3D) → *ansys.geometry.core.designer.designpoint.DesignPoint*

Create a single design point.

Parameters**name**

[*str*] User-defined label for the design points.

points

[Point3D] 3D point constituting the design point.

Component.**add_design_points**(*name: str, points: list[ansys.geometry.core.math.point.Point3D]*) → list[ansys.geometry.core.designer.designpoint.DesignPoint]

Create a list of design points.

Parameters

name

[str] User-defined label for the list of design points.

points

[list[Point3D]] list of the 3D points that constitute the list of design points.

Component.**delete_beam**(*beam: ansys.geometry.core.designer.beam.Beam | str*) → None

Delete an existing beam belonging to this component's scope.

Parameters

beam

[Beam | str] ID of the beam or instance to delete.

Notes

If the beam belongs to this component's children, it is deleted. If the beam does not belong to this component (or its children), it is not deleted.

Component.**search_component**(*id: str*) → Component | None

Search nested components recursively for a component.

Parameters

id

[str] ID of the component to search for.

Returns

Component

Component with the requested ID. If this ID is not found, None is returned.

Component.**search_body**(*id: str*) → ansys.geometry.core.designer.body.Body | None

Search bodies in the component's scope.

Parameters

id

[str] ID of the body to search for.

Returns

Body | None

Body with the requested ID. If the ID is not found, None is returned.

Notes

This method searches for bodies in the component and nested components recursively.

Component.**search_beam**(*id: str*) → ansys.geometry.core.designer.beam.Beam | None

Search beams in the component's scope.

Parameters

id
 [str] ID of the beam to search for.

Returns

Beam | None
 Beam with the requested ID. If the ID is not found, None is returned.

Notes

This method searches for beams in the component and nested components recursively.

Component.**tessellate**(*_recursive_call: bool = False*) → `pyvista.PolyData` | `list[pyvista.MultiBlock]`

Tessellate the component.

Parameters

_recursive_call: bool, default: False
 Internal flag to indicate if this method is being called recursively. Not to be used by the user.

Returns

PolyData, list[MultiBlock]
 Tessellated component as a single PolyData object. If the method is called recursively, a list of MultiBlock objects is returned.

Component.**plot**(*merge_component: bool = True, merge_bodies: bool = True, screenshot: str | None = None, use_trame: bool | None = None, use_service_colors: bool | None = None, allow_picking: bool | None = None, **plotting_options: dict | None*) → `None` | `list[Any]`

Plot the component.

Parameters

merge_component
 [bool, default: True] Whether to merge the component into a single dataset. By default, True. Performance improved. When True, all the faces of the component are effectively merged into a single dataset. If False, the individual bodies are kept separate.

merge_bodies
 [bool, default: True] Whether to merge each body into a single dataset. By default, True. Performance improved. When True, all the faces of each individual body are effectively merged into a single dataset. If False, the individual faces are kept separate.

screenshot
 [str, default: None] Path for saving a screenshot of the image being represented.

use_trame
 [bool, default: None] Whether to enable the use of `trame`. The default is None, in which case the `ansys.tools.visualization_interface.USE_TRAME` global setting is used.

use_service_colors
 [bool, default: None] Whether to use the colors assigned to the body in the service. The default is None, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

allow_picking
 [bool, default: None] Whether to enable picking. The default is None, in which case the picker is not enabled.

****plotting_options**
 [dict, default: None] Keyword arguments for plotting. For allowable keyword arguments, see the

Returns`None | list[Any]`If `allow_picking=True`, a list of picked objects is returned. Otherwise, `None`.**Examples**

Create 25 small cylinders in a grid-like pattern on the XY plane and plot them. Make the cylinders look metallic by enabling physically-based rendering with `pbr=True`.

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> import numpy as np
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 1, 0])
>>> design = modeler.create_design("my-design")
>>> mycomp = design.add_component("my-comp")
>>> n = 5
>>> xx, yy = np.meshgrid(
...     np.linspace(-4, 4, n),
...     np.linspace(-4, 4, n),
... )
>>> for x, y in zip(xx.ravel(), yy.ravel()):
...     sketch = Sketch(plane)
...     sketch.circle(Point2D([x, y]), 0.2 * u.m)
...     mycomp.extrude_sketch(f"body-{x}-{y}", sketch, 1 * u.m)
>>> mycomp
ansys.geometry.core.designer.Component 0x2203cc9ec50
  Name          : my-comp
  Exists        : True
  Parent component : my-design
  N Bodies      : 25
  N Components  : 0
  N Coordinate Systems : 0
>>> mycomp.plot(pbr=True, metallic=1.0)
```

`Component.__repr__() → str`

Represent the Component as a string.

`Component.tree_print(consider_comps: bool = True, consider_bodies: bool = True, consider_beams: bool = True, depth: int | None = None, indent: int = 4, sort_keys: bool = False, return_list: bool = False, skip_loc_header: bool = False) → None | list[str]`

Print the component in tree format.

Parameters**consider_comps**`[bool, default: True]` Whether to print the nested components.**consider_bodies**`[bool, default: True]` Whether to print the bodies.**consider_beams**`[bool, default: True]` Whether to print the beams.

depth

[`int` | `None`, default: `None`] Depth level to print. If `None`, it prints all levels.

indent

[`int`, default: 4] Indentation level. Minimum is 2 - if less than 2, it is set to 2 by default.

sort_keys

[`bool`, default: `False`] Whether to sort the keys alphabetically.

return_list

[`bool`, default: `False`] Whether to return a list of strings or print out the tree structure.

skip_loc_header

[`bool`, default: `False`] Whether to skip the location header. Mostly for internal use.

Returns

`None` | `list[str]`

Tree-style printed component or list of strings representing the component tree.

SharedTopologyType

```
class ansys.geometry.core.designer.component.SharedTopologyType(*args, **kwargs)
```

Bases: `enum.Enum`

Shared topologies available.

Overview**Attributes**

<code>SHARETYPE_NONE</code>
<code>SHARETYPE_SHARE</code>
<code>SHARETYPE_MERGE</code>
<code>SHARETYPE_GROUPS</code>

Import detail

```
from ansys.geometry.core.designer.component import SharedTopologyType
```

Attribute detail

`SharedTopologyType.SHARETYPE_NONE` = 0

`SharedTopologyType.SHARETYPE_SHARE` = 1

`SharedTopologyType.SHARETYPE_MERGE` = 2

`SharedTopologyType.SHARETYPE_GROUPS` = 3

ExtrusionDirection

```
class ansys.geometry.core.designer.component.ExtrusionDirection(*args, **kwargs)
```

Bases: `enum.Enum`

Enum for extrusion direction definition.

Overview

Constructors

<i>from_string</i>	Convert a string to an ExtrusionDirection enum.
--------------------	---

Attributes

<i>POSITIVE</i>
<i>NEGATIVE</i>

Import detail

```
from ansys.geometry.core.designer.component import ExtrusionDirection
```

Attribute detail

ExtrusionDirection.**POSITIVE** = '+'

ExtrusionDirection.**NEGATIVE** = '-'

Method detail

classmethod ExtrusionDirection.**from_string**(*string: str, use_default_if_error: bool = False*) → *ExtrusionDirection*

Convert a string to an ExtrusionDirection enum.

Description

Provides for managing components.

The coordinate_system.py module

Summary

Classes

<i>CoordinateSystem</i>	Represents a user-defined coordinate system within the design assembly.
-------------------------	---

CoordinateSystem

class ansys.geometry.core.designer.coordinate_system.**CoordinateSystem**(*name: str, frame: ansys.geometry.core.math.frame.Frame, parent_component: ansys.geometry.core.designer.component.Component, grpc_client: ansys.geometry.core.connection.client.GrpcClient, preexisting_id: str | None = None*)

Represents a user-defined coordinate system within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

name

[`str`] User-defined label for the coordinate system.

frame

[`Frame`] Frame defining the coordinate system bounds.

parent_component

[`Component`, default: `Component`] Parent component the coordinate system is assigned against.

grpc_client

[`GrpcClient`] Active supporting Geometry service instance for design modeling.

Overview

Properties

<i>id</i>	ID of the coordinate system.
<i>name</i>	Name of the coordinate system.
<i>frame</i>	Frame of the coordinate system.
<i>parent_component</i>	Parent component of the coordinate system.
<i>is_alive</i>	Flag indicating if coordinate system is still alive on the server.

Special methods

<code>__repr__</code>	Represent the coordinate system as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.designer.coordinate_system import CoordinateSystem
```

Property detail

property `CoordinateSystem.id`: `str`

ID of the coordinate system.

property `CoordinateSystem.name`: `str`

Name of the coordinate system.

property `CoordinateSystem.frame`: `ansys.geometry.core.math.frame.Frame`

Frame of the coordinate system.

property `CoordinateSystem.parent_component`:
`ansys.geometry.core.designer.component.Component`

Parent component of the coordinate system.

property `CoordinateSystem.is_alive`: `bool`

Flag indicating if coordinate system is still alive on the server.

Method detail

`CoordinateSystem.__repr__() → str`

Represent the coordinate system as a string.

Description

Provides for managing a user-defined coordinate system.

The design.py module

Summary

Classes

<i>Design</i> Provides for organizing geometry assemblies.
--

Enums

<i>DesignFileFormat</i> Provides supported file formats that can be downloaded for designs.

Design

```
class ansys.geometry.core.designer.design.Design(name: str, modeler:
                                                ansys.geometry.core.modeler.Modeler,
                                                read_existing_design: bool = False)
```

Bases: *ansys.geometry.core.designer.component.Component*

Provides for organizing geometry assemblies.

This class synchronizes to a supporting Geometry service instance.

Parameters

name

[str] User-defined label for the design.

grpc_client

[GrpcClient] Active supporting Geometry service instance for design modeling.

read_existing_design

[bool, default: False] Whether an existing design on the service should be read. This parameter is only valid when connecting to an existing service session. Otherwise, avoid using this optional parameter.

Overview

Methods

<i>close</i>	Close the design.
<i>add_material</i>	Add a material to the design.
<i>save</i>	Save a design to disk on the active Geometry server instance.
<i>download</i>	Export and download the design from the server.
<i>export_to_scdocx</i>	Export the design to an scdocx file.
<i>export_to_disco</i>	Export the design to an disco file.
<i>export_to_stride</i>	Export the design to an stride file.
<i>export_to_parasolid_text</i>	Export the design to a Parasolid text file.
<i>export_to_parasolid_bin</i>	Export the design to a Parasolid binary file.
<i>export_to_fmd</i>	Export the design to an FMD file.
<i>export_to_step</i>	Export the design to a STEP file.
<i>export_to_iges</i>	Export the design to an IGES file.
<i>export_to_pmdb</i>	Export the design to a PMDB file.
<i>create_named_selection</i>	Create a named selection on the active Geometry server instance.
<i>delete_named_selection</i>	Delete a named selection on the active Geometry server instance.
<i>delete_component</i>	Delete a component (itself or its children).
<i>set_shared_topology</i>	Set the shared topology to apply to the component.
<i>add_beam_circular_profile</i>	Add a new beam circular profile under the design for creating beams.
<i>get_all_parameters</i>	Get parameters for the design.
<i>set_parameter</i>	Set or update a parameter of the design.
<i>add_midsurface_thickness</i>	Add a mid-surface thickness to a list of bodies.
<i>add_midsurface_offset</i>	Add a mid-surface offset type to a list of bodies.
<i>delete_beam_profile</i>	Remove a beam profile on the active geometry server instance.
<i>insert_file</i>	Insert a file into the design.

Properties

<i>design_id</i>	The design's object unique id.
<i>materials</i>	List of materials available for the design.
<i>named_selections</i>	List of named selections available for the design.
<i>beam_profiles</i>	List of beam profile available for the design.
<i>parameters</i>	List of parameters available for the design.
<i>is_active</i>	Whether the design is currently active.
<i>is_closed</i>	Whether the design is closed (i.e. not active).

Special methods

<code>__repr__</code>	Represent the Design as a string.
-----------------------	-----------------------------------

Import detail

```
from ansys.geometry.core.designer.design import Design
```

Property detail

property `Design.design_id: str`

The design's object unique id.

property `Design.materials: list[ansys.geometry.core.materials.material.Material]`

List of materials available for the design.

property `Design.named_selections:`
`list[ansys.geometry.core.designer.selection.NamedSelection]`

List of named selections available for the design.

property `Design.beam_profiles: list[ansys.geometry.core.designer.beam.BeamProfile]`

List of beam profile available for the design.

property `Design.parameters: list[ansys.geometry.core.parameters.parameter.Parameter]`

List of parameters available for the design.

property `Design.is_active: bool`

Whether the design is currently active.

property `Design.is_closed: bool`

Whether the design is closed (i.e. not active).

Method detail

`Design.close() → None`

Close the design.

`Design.add_material(material: ansys.geometry.core.materials.material.Material) → None`

Add a material to the design.

Parameters

material

[Material] Material to add.

`Design.save(file_location: pathlib.Path | str) → None`

Save a design to disk on the active Geometry server instance.

Parameters

file_location

[Path | str] Location on disk to save the file to.

`Design.download(file_location: pathlib.Path | str, format: DesignFileFormat = DesignFileFormat.SCDOCX) → None`

Export and download the design from the server.

Parameters

file_location

[Path | str] Location on disk to save the file to.

format

[DesignFileFormat, default: *DesignFileFormat.SDOCX*] Format for the file to save to.

Design.**export_to_sdocx**(location: *pathlib.Path* | *str* | *None* = *None*) → *pathlib.Path*

Export the design to an sdocx file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_disco**(location: *pathlib.Path* | *str* | *None* = *None*) → *pathlib.Path*

Export the design to an disco file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_stride**(location: *pathlib.Path* | *str* | *None* = *None*) → *pathlib.Path*

Export the design to an stride file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_parasolid_text**(location: *pathlib.Path* | *str* | *None* = *None*) → *pathlib.Path*

Export the design to a Parasolid text file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_parasolid_bin**(location: *pathlib.Path* | *str* | *None* = *None*) → *pathlib.Path*

Export the design to a Parasolid binary file.

Parameters

location

[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_fmd**(location: *pathlib.Path* | str | None = None) → *pathlib.Path*

Export the design to an FMD file.

Parameters**location**

[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_step**(location: *pathlib.Path* | str | None = None) → *pathlib.Path*

Export the design to a STEP file.

Parameters**location**

[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_iges**(location: *pathlib.Path* | str = None) → *pathlib.Path*

Export the design to an IGES file.

Parameters**location**

[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

Design.**export_to_pmdb**(location: *pathlib.Path* | str | None = None) → *pathlib.Path*

Export the design to a PMDB file.

Parameters**location**

[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

`Design.create_named_selection`(*name*: *str*, *bodies*: *list*[*ansys.geometry.core.designer.body.Body*] | *None* = *None*, *faces*: *list*[*ansys.geometry.core.designer.face.Face*] | *None* = *None*, *edges*: *list*[*ansys.geometry.core.designer.edge.Edge*] | *None* = *None*, *beams*: *list*[*ansys.geometry.core.designer.beam.Beam*] | *None* = *None*, *design_points*: *list*[*ansys.geometry.core.designer.designpoint.DesignPoint*] | *None* = *None*) → *ansys.geometry.core.designer.selection.NamedSelection*

Create a named selection on the active Geometry server instance.

Parameters

name

[*str*] User-defined name for the named selection.

bodies

[*list*[*Body*], default: *None*] All bodies to include in the named selection.

faces

[*list*[*Face*], default: *None*] All faces to include in the named selection.

edges

[*list*[*Edge*], default: *None*] All edges to include in the named selection.

beams

[*list*[*Beam*], default: *None*] All beams to include in the named selection.

design_points

[*list*[*DesignPoint*], default: *None*] All design points to include in the named selection.

Returns

NamedSelection

Newly created named selection that maintains references to all target entities.

`Design.delete_named_selection`(*named_selection*: *ansys.geometry.core.designer.selection.NamedSelection* | *str*) → *None*

Delete a named selection on the active Geometry server instance.

Parameters

named_selection

[*NamedSelection* | *str*] Name of the named selection or instance.

`Design.delete_component`(*component*: *ansys.geometry.core.designer.component.Component* | *str*) → *None*

Delete a component (itself or its children).

Parameters

id

[*Union*[*Component*, *str*]] Name of the component or instance to delete.

Raises

ValueError

The design itself cannot be deleted.

Notes

If the component is not this component (or its children), it is not deleted.

`Design.set_shared_topology`(*share_type*: *ansys.geometry.core.designer.component.SharedTopologyType*) → *None*

Set the shared topology to apply to the component.

Parameters**share_type**

[SharedTopologyType] Shared topology type to assign.

Raises**ValueError**

Shared topology does not apply to a design.

`Design.add_beam_circular_profile(name: str, radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance, center: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D = ZERO_POINT3D, direction_x: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = UNITVECTOR3D_X, direction_y: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = UNITVECTOR3D_Y) → ansys.geometry.core.designer.beam.BeamCircularProfile`

Add a new beam circular profile under the design for creating beams.

Parameters**name**

[str] User-defined label for the new beam circular profile.

radius

[Quantity | Distance] Radius of the beam circular profile.

center

[ndarray | RealSequence | Point3D] Center of the beam circular profile.

direction_x

[ndarray | RealSequence | UnitVector3D | Vector3D] X-plane direction.

direction_y

[ndarray | RealSequence | UnitVector3D | Vector3D] Y-plane direction.

`Design.get_all_parameters() → list[ansys.geometry.core.parameters.parameter.Parameter]`

Get parameters for the design.

Returns**list[Parameter]**

List of parameters for the design.

`Design.set_parameter(dimension: ansys.geometry.core.parameters.parameter.Parameter) → ansys.geometry.core.parameters.parameter.ParameterUpdateStatus`

Set or update a parameter of the design.

Parameters**dimension**

[Parameter] Parameter to set.

Returns**ParameterUpdateStatus**

Status of the update operation.

`Design.add_midsurface_thickness`(*thickness*: *pint.Quantity*, *bodies*:
list[ansys.geometry.core.designer.body.Body]) → *None*

Add a mid-surface thickness to a list of bodies.

Parameters

thickness

[*Quantity*] Thickness to be assigned.

bodies

[*list[Body]*] All bodies to include in the mid-surface thickness assignment.

Notes

Only surface bodies will be eligible for mid-surface thickness assignment.

`Design.add_midsurface_offset`(*offset_type*: *ansys.geometry.core.designer.body.MidSurfaceOffsetType*, *bodies*:
list[ansys.geometry.core.designer.body.Body]) → *None*

Add a mid-surface offset type to a list of bodies.

Parameters

offset_type

[*MidSurfaceOffsetType*] Surface offset to be assigned.

bodies

[*list[Body]*] All bodies to include in the mid-surface offset assignment.

Notes

Only surface bodies will be eligible for mid-surface offset assignment.

`Design.delete_beam_profile`(*beam_profile*: *ansys.geometry.core.designer.beam.BeamProfile | str*) → *None*

Remove a beam profile on the active geometry server instance.

Parameters

beam_profile

[*BeamProfile | str*] A beam profile name or instance that should be deleted.

`Design.insert_file`(*file_location*: *pathlib.Path | str*, *import_options*:
ansys.geometry.core.misc.options.ImportOptions = ImportOptions()) →
ansys.geometry.core.designer.component.Component

Insert a file into the design.

Parameters

file_location

[*Path | str*] Location on disk where the file is located.

import_options

[*ImportOptions*] The options to pass into upload file

Returns

Component

The newly inserted component.

`Design.__repr__`() → *str*

Represent the Design as a string.

DesignFileFormat

class ansys.geometry.core.designer.design.DesignFileFormat(*args, **kwargs)

Bases: `enum.Enum`

Provides supported file formats that can be downloaded for designs.

Overview

Attributes

<i>SCDOCX</i>
<i>PARASOLID_TEXT</i>
<i>PARASOLID_BIN</i>
<i>FMD</i>
<i>STEP</i>
<i>IGES</i>
<i>PMDB</i>
<i>STRIDE</i>
<i>DISCO</i>
<i>INVALID</i>

Import detail

```
from ansys.geometry.core.designer.design import DesignFileFormat
```

Attribute detail

`DesignFileFormat.SCDOCX`

`DesignFileFormat.PARASOLID_TEXT`

`DesignFileFormat.PARASOLID_BIN`

`DesignFileFormat.FMD`

`DesignFileFormat.STEP`

`DesignFileFormat.IGES`

`DesignFileFormat.PMDB`

`DesignFileFormat.STRIDE`

`DesignFileFormat.DISCO`

`DesignFileFormat.INVALID = ('INVALID', None)`

Description

Provides for managing designs.

The designpoint.py module

Summary

Classes

<i>DesignPoint</i>	Provides for creating design points in components.
--------------------	--

DesignPoint

```
class ansys.geometry.core.designer.designpoint.DesignPoint(id: str, name: str, point: ansys.geometry.core.math.point.Point3D, parent_component: ansys.geometry.core.designer.component.Component)
```

Provides for creating design points in components.

Parameters

id

[str] Server-defined ID for the design points.

name

[str] User-defined label for the design points.

points

[Point3D] 3D point constituting the design points.

parent_component

[Component] Parent component to place the new design point under within the design assembly.

Overview

Properties

<i>id</i>	ID of the design point.
<i>name</i>	Name of the design point.
<i>value</i>	Value of the design point.
<i>parent_component</i>	Component node that the design point is under.

Special methods

<code>__repr__</code>	Represent the design points as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.designer.designpoint import DesignPoint
```

Property detail

property `DesignPoint.id: str`

ID of the design point.

property `DesignPoint.name: str`

Name of the design point.

property `DesignPoint.value: ansys.geometry.core.math.point.Point3D`

Value of the design point.

property `DesignPoint.parent_component: ansys.geometry.core.designer.component.Component`

Component node that the design point is under.

Method detail

`DesignPoint.__repr__() → str`

Represent the design points as a string.

Description

Module for creating and managing design points.

The edge.py module

Summary

Classes

<i>Edge</i> Represents a single edge of a body within the design assembly.
--

Enums

<i>CurveType</i> Provides values for the curve types supported.

Edge

```
class ansys.geometry.core.designer.edge.Edge(id: str, curve_type: CurveType, body:  
ansys.geometry.core.designer.body.Body, grpc_client:  
ansys.geometry.core.connection.client.GrpcClient,  
is_reversed: bool = False)
```

Represents a single edge of a body within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

id

[str] Server-defined ID for the body.

curve_type

[CurveType] Type of curve that the edge forms.

body

[Body] Parent body that the edge constructs.

grpc_client

[GrpcClient] Active supporting Geometry service instance for design modeling.

is_reversed

[bool] Direction of the edge.

Overview**Properties**

<i>id</i>	ID of the edge.
<i>body</i>	Body of the edge.
<i>is_reversed</i>	Flag indicating if the edge is reversed.
<i>shape</i>	Underlying trimmed curve of the edge.
<i>length</i>	Calculated length of the edge.
<i>curve_type</i>	Curve type of the edge.
<i>faces</i>	Faces that contain the edge.
<i>start</i>	Start point of the edge.
<i>end</i>	End point of the edge.

Import detail

```
from ansys.geometry.core.designer.edge import Edge
```

Property detail

property Edge.*id*: **str**

ID of the edge.

property Edge.*body*: **ansys.geometry.core.designer.body.Body**

Body of the edge.

property Edge.*is_reversed*: **bool**

Flag indicating if the edge is reversed.

property Edge.*shape*: **ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve**

Underlying trimmed curve of the edge.

If the edge is reversed, its shape is the ReversedTrimmedCurve type, which swaps the start and end points of the curve and handles parameters to allow evaluation as if the curve is not reversed.

property Edge.*length*: **pint.Quantity**

Calculated length of the edge.

property Edge.*curve_type*: **CurveType**

Curve type of the edge.

property Edge.*faces*: **list[ansys.geometry.core.designer.face.Face]**

Faces that contain the edge.

property Edge.*start*: **ansys.geometry.core.math.point.Point3D**

Start point of the edge.

property Edge.*end*: **ansys.geometry.core.math.point.Point3D**

End point of the edge.

CurveType

class ansys.geometry.core.designer.edge.CurveType(*args, **kwargs)

Bases: `enum.Enum`

Provides values for the curve types supported.

Overview

Attributes

<code>CURVETYPE_UNKNOWN</code>
<code>CURVETYPE_LINE</code>
<code>CURVETYPE_CIRCLE</code>
<code>CURVETYPE_ELLIPSE</code>
<code>CURVETYPE_NURBS</code>
<code>CURVETYPE_PROCEDURAL</code>

Import detail

```
from ansys.geometry.core.designer.edge import CurveType
```

Attribute detail

`CurveType.CURVETYPE_UNKNOWN = 0`

`CurveType.CURVETYPE_LINE = 1`

`CurveType.CURVETYPE_CIRCLE = 2`

`CurveType.CURVETYPE_ELLIPSE = 3`

`CurveType.CURVETYPE_NURBS = 4`

`CurveType.CURVETYPE_PROCEDURAL = 5`

Description

Module for managing an edge.

The face.py module

Summary

Classes

<code>FaceLoop</code>	Provides an internal class holding the face loops defined.
<code>Face</code>	Represents a single face of a body within the design assembly.

Enums

<i>SurfaceType</i>	Provides values for the surface types supported.
<i>FaceLoopType</i>	Provides values for the face loop types supported.

FaceLoop

```
class ansys.geometry.core.designer.face.FaceLoop(type: FaceLoopType, length: pint.Quantity,
                                                min_bbox: ansys.geometry.core.math.point.Point3D,
                                                max_bbox: ansys.geometry.core.math.point.Point3D,
                                                edges:
                                                list[ansys.geometry.core.designer.edge.Edge])
```

Provides an internal class holding the face loops defined.

Parameters

type	[FaceLoopType] Type of loop.
length	[Quantity] Length of the loop.
min_bbox	[Point3D] Minimum point of the bounding box containing the loop.
max_bbox	[Point3D] Maximum point of the bounding box containing the loop.
edges	[list[Edge]] Edges contained in the loop.

Notes

This class is to be used only when parsing server side results. It is not intended to be instantiated by a user.

Overview

Properties

<i>type</i>	Type of the loop.
<i>length</i>	Length of the loop.
<i>min_bbox</i>	Minimum point of the bounding box containing the loop.
<i>max_bbox</i>	Maximum point of the bounding box containing the loop.
<i>edges</i>	Edges contained in the loop.

Import detail

```
from ansys.geometry.core.designer.face import FaceLoop
```

Property detail

property FaceLoop.**type**: *FaceLoopType*
Type of the loop.

property FaceLoop.length: `pint.Quantity`

Length of the loop.

property FaceLoop.min_bbox: `ansys.geometry.core.math.point.Point3D`

Minimum point of the bounding box containing the loop.

property FaceLoop.max_bbox: `ansys.geometry.core.math.point.Point3D`

Maximum point of the bounding box containing the loop.

property FaceLoop.edges: `list[ansys.geometry.core.designer.edge.Edge]`

Edges contained in the loop.

Face

class `ansys.geometry.core.designer.face.Face`(*id*: `str`, *surface_type*: `SurfaceType`, *body*: `ansys.geometry.core.designer.body.Body`, *grpc_client*: `ansys.geometry.core.connection.client.GrpcClient`, *is_reversed*: `bool = False`)

Represents a single face of a body within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

id

[`str`] Server-defined ID for the body.

surface_type

[`SurfaceType`] Type of surface that the face forms.

body

[`Body`] Parent body that the face constructs.

grpc_client

[`GrpcClient`] Active supporting Geometry service instance for design modeling.

Overview

Methods

<code>set_color</code>	Set the color of the face.
<code>normal</code>	Get the normal direction to the face at certain UV coordinates.
<code>face_normal</code>	Get the normal direction to the face at certain UV coordinates.
<code>point</code>	Get a point of the face evaluated at certain UV coordinates.
<code>face_point</code>	Get a point of the face evaluated at certain UV coordinates.
<code>create_isoparametric_curves</code>	Create isoparametric curves at the given proportional parameter.
<code>setup_offset_relationship</code>	Create an offset relationship between two faces.
<code>tessellate</code>	Tessellate the face and return the geometry as triangles.
<code>plot</code>	Plot the face.

Properties

<i>id</i>	Face ID.
<i>is_reversed</i>	Flag indicating if the face is reversed.
<i>body</i>	Body that the face belongs to.
<i>shape</i>	Underlying trimmed surface of the face.
<i>surface_type</i>	Surface type of the face.
<i>area</i>	Calculated area of the face.
<i>edges</i>	List of all edges of the face.
<i>loops</i>	List of all loops of the face.
<i>color</i>	Get the current color of the face.
<i>bounding_box</i>	Get the bounding box for the face.

Import detail

```
from ansys.geometry.core.designer.face import Face
```

Property detail

property `Face.id: str`

Face ID.

property `Face.is_reversed: bool`

Flag indicating if the face is reversed.

property `Face.body: ansys.geometry.core.designer.body.Body`

Body that the face belongs to.

property `Face.shape: ansys.geometry.core.shapes-surfaces-trimmed-surface.TrimmedSurface`

Underlying trimmed surface of the face.

If the face is reversed, its shape is a `ReversedTrimmedSurface` type, which handles the direction of the normal vector to ensure it is always facing outward.

property `Face.surface_type: SurfaceType`

Surface type of the face.

property `Face.area: pint.Quantity`

Calculated area of the face.

property `Face.edges: list[ansys.geometry.core.designer.edge.Edge]`

List of all edges of the face.

property `Face.loops: list[FaceLoop]`

List of all loops of the face.

property `Face.color: str`

Get the current color of the face.

property `Face.bounding_box: ansys.geometry.core.math.bbox.BoundingBox2D`

Get the bounding box for the face.

Method detail

Face.**set_color**(color: *str* | *tuple*[float, float, float]) → None

Set the color of the face.

Face.**normal**(u: float = 0.5, v: float = 0.5) → *ansys.geometry.core.math.vector.UnitVector3D*

Get the normal direction to the face at certain UV coordinates.

Parameters

u

[float, default: 0.5] First coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

v

[float, default: 0.5] Second coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

Returns

UnitVector3D

UnitVector3D object evaluated at the given U and V coordinates. This UnitVector3D object is perpendicular to the surface at the given UV coordinates.

Notes

To properly use this method, you must handle UV coordinates. Thus, you must know how these relate to the underlying Geometry service. It is an advanced method for Geometry experts only.

Face.**face_normal**(u: float = 0.5, v: float = 0.5) → *ansys.geometry.core.math.vector.UnitVector3D*

Get the normal direction to the face at certain UV coordinates.

Parameters

u

[float, default: 0.5] First coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

v

[float, default: 0.5] Second coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

Returns

UnitVector3D

UnitVector3D object evaluated at the given U and V coordinates. This UnitVector3D object is perpendicular to the surface at the given UV coordinates.

Notes

This method is deprecated. Use the `normal` method instead.

Face.**point**(u: float = 0.5, v: float = 0.5) → *ansys.geometry.core.math.point.Point3D*

Get a point of the face evaluated at certain UV coordinates.

Parameters

u

[float, default: 0.5] First coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

v
 [float, default: 0.5] Second coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

Returns

Point3D
 Point3D object evaluated at the given UV coordinates.

Notes

To properly use this method, you must handle UV coordinates. Thus, you must know how these relate to the underlying Geometry service. It is an advanced method for Geometry experts only.

Face.**face_point**(*u*: float = 0.5, *v*: float = 0.5) → *ansys.geometry.core.math.point.Point3D*

Get a point of the face evaluated at certain UV coordinates.

Parameters

u
 [float, default: 0.5] First coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

v
 [float, default: 0.5] Second coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

Returns

Point3D
 Point3D object evaluated at the given UV coordinates.

Notes

This method is deprecated. Use the `point` method instead.

Face.**create_isoparametric_curves**(*use_u_param*: bool, *parameter*: float) →
list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]

Create isoparametric curves at the given proportional parameter.

Typically, only one curve is created, but if the face has a hole, it is possible that more than one curve is created.

Parameters

use_u_param
 [bool] Whether the parameter is the u coordinate or v coordinate. If True, it is the u coordinate. If False, it is the v coordinate.

parameter
 [float] Proportional [0-1] parameter to create the one or more curves at.

Returns

list[TrimmedCurve]
 list of curves that were created.

Face.**setup_offset_relationship**(*other_face*: Face, *set_baselines*: bool = False, *process_adjacent_faces*: bool = False) → bool

Create an offset relationship between two faces.

Parameters

other_face

[Face] The face to setup an offset relationship with.

set_baselines

[bool, default: `False`] Automatically set baseline faces.

process_adjacent_faces

[bool, default: `False`] Look for relationships of the same offset on adjacent faces.

Returns**bool**

True when successful, False when failed.

Face.**tessellate**() → `pyvista.PolyData`

Tessellate the face and return the geometry as triangles.

Returns**PolyData**

`pyvista.PolyData` object holding the face.

Face.**plot**(*screenshot*: `str` | `None` = `None`, *use_frame*: `bool` | `None` = `None`, *use_service_colors*: `bool` | `None` = `None`, ***plotting_options*: `dict` | `None`) → `None`

Plot the face.

Parameters**screenshot**

[`str`, default: `None`] Path for saving a screenshot of the image that is being represented.

use_frame

[`bool`, default: `None`] Whether to enable the use of `frame`. The default is `None`, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.

use_service_colors

[`bool`, default: `None`] Whether to use the colors assigned to the face in the service. The default is `None`, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

****plotting_options**

[`dict`, default: `None`] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

SurfaceType

`class` `ansys.geometry.core.designer.face.SurfaceType(*args, **kws)`

Bases: `enum.Enum`

Provides values for the surface types supported.

Overview

Attributes

<code>SURFACTYPE_UNKNOWN</code>
<code>SURFACTYPE_PLANE</code>
<code>SURFACTYPE_CYLINDER</code>
<code>SURFACTYPE_CONE</code>
<code>SURFACTYPE_TORUS</code>
<code>SURFACTYPE_SPHERE</code>
<code>SURFACTYPE_NURBS</code>
<code>SURFACTYPE_PROCEDURAL</code>

Import detail

```
from ansys.geometry.core.designer.face import SurfaceType
```

Attribute detail

```
SurfaceType.SURFACTYPE_UNKNOWN = 0
SurfaceType.SURFACTYPE_PLANE = 1
SurfaceType.SURFACTYPE_CYLINDER = 2
SurfaceType.SURFACTYPE_CONE = 3
SurfaceType.SURFACTYPE_TORUS = 4
SurfaceType.SURFACTYPE_SPHERE = 5
SurfaceType.SURFACTYPE_NURBS = 6
SurfaceType.SURFACTYPE_PROCEDURAL = 7
```

FaceLoopType

```
class ansys.geometry.core.designer.face.FaceLoopType(*args, **kwargs)
```

Bases: `enum.Enum`

Provides values for the face loop types supported.

Overview

Attributes

<code>INNER_LOOP</code>
<code>OUTER_LOOP</code>

Import detail

```
from ansys.geometry.core.designer.face import FaceLoopType
```

Attribute detail

FaceLoopType.INNER_LOOP = 'INNER'

FaceLoopType.OUTER_LOOP = 'OUTER'

Description

Module for managing a face.

The geometry_commands.py module

Summary

Classes

<i>GeometryCommands</i>	Provides geometry commands for PyAnsys Geometry.
-------------------------	--

Enums

<i>ExtrudeType</i>	Provides values for extrusion types.
<i>OffsetMode</i>	Provides values for offset modes during extrusions.
<i>FillPatternType</i>	Provides values for types of fill patterns.

GeometryCommands

class ansys.geometry.core.designer.geometry_commands.**GeometryCommands**(*grpc_client*: ansys.geometry.core.connection.client.GrpcClient)

Provides geometry commands for PyAnsys Geometry.

Parameters

grpc_client

[GrpcClient] gRPC client to use for the geometry commands.

Overview

Methods

<code>chamfer</code>	Create a chamfer on an edge or adjust the chamfer of a face.
<code>fillet</code>	Create a fillet on an edge or adjust the fillet of a face.
<code>full_fillet</code>	Create a full fillet between a collection of faces.
<code>extrude_faces</code>	Extrude a selection of faces.
<code>extrude_faces_up_to</code>	Extrude a selection of faces up to another object.
<code>extrude_edges</code>	Extrude a selection of edges. Provide either a face or a direction and point.
<code>extrude_edges_up_to</code>	Extrude a selection of edges up to another object.
<code>rename_object</code>	Rename an object.
<code>create_linear_pattern</code>	Create a linear pattern. The pattern can be one or two dimensions.
<code>modify_linear_pattern</code>	Modify a linear pattern. Leave an argument at 0 for it to remain unchanged.
<code>create_circular_pattern</code>	Create a circular pattern. The pattern can be one or two dimensions.
<code>modify_circular_pattern</code>	Modify a circular pattern. Leave an argument at 0 for it to remain unchanged.
<code>create_fill_pattern</code>	Create a fill pattern.
<code>update_fill_pattern</code>	Update a fill pattern.
<code>revolve_faces</code>	Revolve face around an axis.
<code>revolve_faces_up_to</code>	Revolve face around an axis up to a certain object.
<code>revolve_faces_by_helix</code>	Revolve face around an axis in a helix shape.
<code>replace_face</code>	Replace a face with another face.
<code>split_body</code>	Split bodies with a plane, slicers, or faces.
<code>get_round_info</code>	Get info on the rounding of a face.

Import detail

```
from ansys.geometry.core.designer.geometry_commands import GeometryCommands
```

Method detail

`GeometryCommands.chamfer` (*selection*: `ansys.geometry.core.designer.edge.Edge` | `list`[`ansys.geometry.core.designer.edge.Edge`] | `ansys.geometry.core.designer.face.Face` | `list`[`ansys.geometry.core.designer.face.Face`], *distance*: `ansys.geometry.core.typing.Real`) → `bool`

Create a chamfer on an edge or adjust the chamfer of a face.

Parameters

selection

[`Edge` | `list`[`Edge`] | `Face` | `list`[`Face`]] One or more edges or faces to act on.

distance

[`Real`] Chamfer distance.

Returns

bool

True when successful, False when failed.

`GeometryCommands.fillet` (*selection*: `ansys.geometry.core.designer.edge.Edge` | `list`[`ansys.geometry.core.designer.edge.Edge`] | `ansys.geometry.core.designer.face.Face` | `list`[`ansys.geometry.core.designer.face.Face`], *radius*: `ansys.geometry.core.typing.Real`) → `bool`

Create a fillet on an edge or adjust the fillet of a face.

Parameters

selection

[Edge | list[Edge] | Face | list[Face]] One or more edges or faces to act on.

radius

[Real] Fillet radius.

Returns

bool

True when successful, False when failed.

GeometryCommands.**full_fillet**(*faces*: list[ansys.geometry.core.designer.face.Face]) → bool

Create a full fillet between a collection of faces.

Parameters

faces

[list[Face]] Faces to round.

Returns

bool

True when successful, False when failed.

GeometryCommands.**extrude_faces**(*faces*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *distance*: ansys.geometry.core.typing.Real, *direction*: ansys.geometry.core.math.vector.UnitVector3D = None, *extrude_type*: ExtrudeType = ExtrudeType.ADD, *offset_mode*: OffsetMode = OffsetMode.MOVE_FACES_TOGETHER, *pull_symmetric*: bool = False, *copy*: bool = False, *force_do_as_extrude*: bool = False) → list[ansys.geometry.core.designer.body.Body]

Extrude a selection of faces.

Parameters

faces

[Face | list[Face]] Faces to extrude.

distance

[Real] Distance to extrude.

direction

[UnitVector3D, default: None] Direction of extrusion. If no direction is provided, it will be inferred.

extrude_type

[ExtrudeType, default: ExtrudeType.ADD] Type of extrusion to be performed.

offset_mode

[OffsetMode, default: OffsetMode.MOVE_FACES_TOGETHER] Mode of how to handle offset relationships.

pull_symmetric

[bool, default: False] Pull symmetrically on both sides if True.

copy

[bool, default: False] Copy the face and move it instead of extruding the original face if True.

force_do_as_extrude

[bool, default: `False`] Forces to do as an extrusion if `True`, if `False` allows extrusion by offset.

Returns**list[Body]**

Bodies created by the extrusion if any.

```
GeometryCommands.extrude_faces_up_to(faces: ansys.geometry.core.designer.face.Face |
    list[ansys.geometry.core.designer.face.Face], up_to_selection:
    ansys.geometry.core.designer.face.Face |
    ansys.geometry.core.designer.edge.Edge |
    ansys.geometry.core.designer.body.Body, seed_point:
    ansys.geometry.core.math.point.Point3D, direction:
    ansys.geometry.core.math.vector.UnitVector3D, extrude_type:
    ExtrudeType = ExtrudeType.ADD, offset_mode: OffsetMode =
    OffsetMode.MOVE_FACES_TOGETHER, pull_symmetric: bool =
    False, copy: bool = False, force_do_as_extrude: bool = False) →
    list[ansys.geometry.core.designer.body.Body]
```

Extrude a selection of faces up to another object.

Parameters**faces**

[Face | list[Face]] Faces to extrude.

up_to_selection

[Face | Edge | Body] The object to pull the faces up to.

seed_point

[Point3D] Origin to define the extrusion.

direction

[UnitVector3D, default: `None`] Direction of extrusion. If no direction is provided, it will be inferred.

extrude_type

[ExtrudeType, default: `ExtrudeType.ADD`] Type of extrusion to be performed.

offset_mode

[OffsetMode, default: `OffsetMode.MOVE_FACES_TOGETHER`] Mode of how to handle offset relationships.

pull_symmetric

[bool, default: `False`] Pull symmetrically on both sides if `True`.

copy

[bool, default: `False`] Copy the face and move it instead of extruding the original face if `True`.

force_do_as_extrude

[bool, default: `False`] Forces to do as an extrusion if `True`, if `False` allows extrusion by offset.

Returns**list[Body]**

Bodies created by the extrusion if any.

GeometryCommands.**extrude_edges**(*edges*: ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge], *distance*: ansys.geometry.core.typing.Real, *from_face*: ansys.geometry.core.designer.face.Face = None, *from_point*: ansys.geometry.core.math.point.Point3D = None, *direction*: ansys.geometry.core.math.vector.UnitVector3D = None, *extrude_type*: ExtrudeType = ExtrudeType.ADD, *pull_symmetric*: bool = False, *copy*: bool = False, *natural_extension*: bool = False) → *list*[ansys.geometry.core.designer.body.Body]

Extrude a selection of edges. Provide either a face or a direction and point.

Parameters

edges

[Edge | *list*[Edge]] Edges to extrude.

distance

[Real] Distance to extrude.

from_face

[Face, default: None] Face to pull normal from.

from_point

[Point3D, default: None] Point to pull from. Must be used with *direction*.

direction

[UnitVector3D, default: None] Direction to pull. Must be used with *from_point*.

extrude_type

[ExtrudeType, default: ExtrudeType.ADD] Type of extrusion to be performed.

pull_symmetric

[bool, default: False] Pull symmetrically on both sides if True.

copy

[bool, default: False] Copy the edge and move it instead of extruding the original edge if True.

natural_extension

[bool, default: False] Surfaces will extend in a natural or linear shape after exceeding its original range.

Returns

list[Body]

Bodies created by the extrusion if any.

GeometryCommands.**extrude_edges_up_to**(*edges*: ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge], *up_to_selection*: ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.body.Body, *seed_point*: ansys.geometry.core.math.point.Point3D, *direction*: ansys.geometry.core.math.vector.UnitVector3D, *extrude_type*: ExtrudeType = ExtrudeType.ADD) → *list*[ansys.geometry.core.designer.body.Body]

Extrude a selection of edges up to another object.

Parameters

edges

[Edge | list[Edge]] Edges to extrude.

up_to_selection

[Face, default: None] The object to pull the faces up to.

seed_point

[Point3D] Origin to define the extrusion.

direction

[UnitVector3D, default: None] Direction of extrusion.

extrude_type

[ExtrudeType, default: ExtrudeType.ADD] Type of extrusion to be performed.

Returns

list[Body]

Bodies created by the extrusion if any.

GeometryCommands.**rename_object**(*selection: list[ansys.geometry.core.designer.body.Body] | list[ansys.geometry.core.designer.component.Component] | list[ansys.geometry.core.designer.face.Face] | list[ansys.geometry.core.designer.edge.Edge], name: str*) → bool

Rename an object.

Parameters

selection

[list[Body] | list[Component] | list[Face] | list[Edge]] Selection of the object to rename.

name

[str] New name for the object.

Returns

bool

True when successful, False when failed.

GeometryCommands.**create_linear_pattern**(*selection: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], linear_direction: ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.face.Face, count_x: int, pitch_x: ansys.geometry.core.typing.Real, two_dimensional: bool = False, count_y: int = None, pitch_y: ansys.geometry.core.typing.Real = None*) → bool

Create a linear pattern. The pattern can be one or two dimensions.

Parameters

selection

[Face | list[Face]] Faces to create the pattern out of.

linear_direction

[Edge | Face] Direction of the linear pattern, determined by the direction of an edge or face normal.

count_x

[int] How many times the pattern repeats in the x direction.

pitch_x

[Real] The spacing between each pattern member in the x direction.

two_dimensional

[bool, default: False] If True, create a pattern in the x and y direction.

count_y

[int, default: None] How many times the pattern repeats in the y direction.

pitch_y

[Real, default: None] The spacing between each pattern member in the y direction.

Returns**bool**

True when successful, False when failed.

GeometryCommands.**modify_linear_pattern**(*selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *count_x*: int = 0, *pitch_x*: ansys.geometry.core.typing.Real = 0.0, *count_y*: int = 0, *pitch_y*: ansys.geometry.core.typing.Real = 0.0, *new_seed_index*: int = 0, *old_seed_index*: int = 0) → bool

Modify a linear pattern. Leave an argument at 0 for it to remain unchanged.

Parameters**selection**

[Face | list[Face]] Faces that belong to the pattern.

count_x

[int, default: 0] How many times the pattern repeats in the x direction.

pitch_x

[Real, default: 0.0] The spacing between each pattern member in the x direction.

count_y

[int, default: 0] How many times the pattern repeats in the y direction.

pitch_y

[Real, default: 0.0] The spacing between each pattern member in the y direction.

new_seed_index

[int, default: 0] The new seed index of the member.

old_seed_index

[int, default: 0] The old seed index of the member.

Returns**bool**

True when successful, False when failed.

GeometryCommands.**create_circular_pattern**(*selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *circular_axis*: ansys.geometry.core.designer.edge.Edge, *circular_count*: int, *circular_angle*: ansys.geometry.core.typing.Real, *two_dimensional*: bool = False, *linear_count*: int = None, *linear_pitch*: ansys.geometry.core.typing.Real = None, *radial_direction*: ansys.geometry.core.math.vector.UnitVector3D = None) → bool

Create a circular pattern. The pattern can be one or two dimensions.

Parameters**selection**

[Face | list[Face]] Faces to create the pattern out of.

circular_axis

[Edge] The axis of the circular pattern, determined by the direction of an edge.

circular_count

[int] How many members are in the circular pattern.

circular_angle

[Real] The angular range of the pattern.

two_dimensional

[bool, default: False] If True, create a two-dimensional pattern.

linear_count

[int, default: None] How many times the circular pattern repeats along the radial lines for a two-dimensional pattern.

linear_pitch

[Real, default: None] The spacing along the radial lines for a two-dimensional pattern.

radial_direction

[UnitVector3D, default: None] The direction from the center out for a two-dimensional pattern.

Returns**bool**

True when successful, False when failed.

GeometryCommands.**modify_circular_pattern**(*selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *circular_count*: int = 0, *linear_count*: int = 0, *step_angle*: ansys.geometry.core.typing.Real = 0.0, *step_linear*: ansys.geometry.core.typing.Real = 0.0) → bool

Modify a circular pattern. Leave an argument at 0 for it to remain unchanged.

Parameters**selection**

[Face | list[Face]] Faces that belong to the pattern.

circular_count

[int, default: 0] How many members are in the circular pattern.

linear_count

[int, default: 0] How many times the circular pattern repeats along the radial lines for a two-dimensional pattern.

step_angle

[Real, default: 0.0] Defines the circular angle.

step_linear

[Real, default: 0.0] Defines the step, along the radial lines, for a pattern dimension greater than 1.

Returns**bool**

True when successful, False when failed.

```
GeometryCommands.create_fill_pattern(selection: ansys.geometry.core.designer.face.Face |
                                     list[ansys.geometry.core.designer.face.Face], linear_direction:
                                     ansys.geometry.core.designer.edge.Edge |
                                     ansys.geometry.core.designer.face.Face, fill_pattern_type:
                                     FillPatternType, margin: ansys.geometry.core.typing.Real,
                                     x_spacing: ansys.geometry.core.typing.Real, y_spacing:
                                     ansys.geometry.core.typing.Real, row_x_offset:
                                     ansys.geometry.core.typing.Real = 0, row_y_offset:
                                     ansys.geometry.core.typing.Real = 0, column_x_offset:
                                     ansys.geometry.core.typing.Real = 0, column_y_offset:
                                     ansys.geometry.core.typing.Real = 0) → bool
```

Create a fill pattern.

Parameters

selection

[Face | list[Face]] Faces to create the pattern out of.

linear_direction

[Edge] Direction of the linear pattern, determined by the direction of an edge.

fill_pattern_type

[FillPatternType] The type of fill pattern.

margin

[Real] Margin defining the border of the fill pattern.

x_spacing

[Real] Spacing between the pattern members in the x direction.

y_spacing

[Real] Spacing between the pattern members in the x direction.

row_x_offset

[Real, default: 0] Offset for the rows in the x direction. Only used with FillPattern.SKEWED.

row_y_offset

[Real, default: 0] Offset for the rows in the y direction. Only used with FillPattern.SKEWED.

column_x_offset

[Real, default: 0] Offset for the columns in the x direction. Only used with FillPattern.SKEWED.

column_y_offset

[Real, default: 0] Offset for the columns in the y direction. Only used with FillPattern.SKEWED.

Returns

bool

True when successful, False when failed.

```
GeometryCommands.update_fill_pattern(selection: ansys.geometry.core.designer.face.Face |
                                      list[ansys.geometry.core.designer.face.Face]) → bool
```

Update a fill pattern.

When the face that a fill pattern exists upon changes in size, the fill pattern can be updated to fill the new space.

Parameters

selection

[Face | `list`[Face]] Face(s) that are part of a fill pattern.

Returns**bool**

True when successful, False when failed.

GeometryCommands.**revolve_faces**(*selection*: ansys.geometry.core.designer.face.Face | `list`[ansys.geometry.core.designer.face.Face], *axis*: ansys.geometry.core.shapes.curves.line.Line, *angle*: ansys.geometry.core.typing.Real) → `list`[ansys.geometry.core.designer.body.Body]

Revolve face around an axis.

Parameters**selection**

[Face | `list`[Face]] Face(s) to revolve.

axis

[Line] Axis of revolution.

angle

[Real] Angular distance to revolve.

Returns**`list`[Body]**

Bodies created by the extrusion if any.

GeometryCommands.**revolve_faces_up_to**(*selection*: ansys.geometry.core.designer.face.Face | `list`[ansys.geometry.core.designer.face.Face], *up_to*: ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.body.Body, *axis*: ansys.geometry.core.shapes.curves.line.Line, *direction*: ansys.geometry.core.math.vector.UnitVector3D, *extrude_type*: ExtrudeType = *ExtrudeType*.ADD) → `list`[ansys.geometry.core.designer.body.Body]

Revolve face around an axis up to a certain object.

Parameters**selection**

[Face | `list`[Face]] Face(s) to revolve.

up_to

[Face | Edge | Body] Object to revolve the face up to.

axis

[Line] Axis of revolution.

direction

[UnitVector3D] Direction of extrusion.

extrude_type

[ExtrudeType, default: *ExtrudeType*.ADD] Type of extrusion to be performed.

Returns

list[Body]

Bodies created by the extrusion if any.

GeometryCommands.**revolve_faces_by_helix**(*selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *axis*: ansys.geometry.core.shapes.curves.line.Line, *direction*: ansys.geometry.core.math.vector.UnitVector3D, *height*: ansys.geometry.core.typing.Real, *pitch*: ansys.geometry.core.typing.Real, *taper_angle*: ansys.geometry.core.typing.Real, *right_handed*: bool, *both_sides*: bool) → list[ansys.geometry.core.designer.body.Body]

Revolve face around an axis in a helix shape.

Parameters**selection**

[Face | list[Face]] Face(s) to revolve.

axis

[Line] Axis of revolution.

direction

[UnitVector3D] Direction of extrusion.

height

[Real,] Height of the helix.

pitch

[Real,] Pitch of the helix.

taper_angle

[Real,] Tape angle of the helix.

right_handed

[bool,] Right-handed helix if True, left-handed if False.

both_sides

[bool,] Create on both sides if True, one side if False.

Returns**list[Body]**

Bodies created by the extrusion if any.

GeometryCommands.**replace_face**(*target_selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *replacement_selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face]) → bool

Replace a face with another face.

Parameters**target_selection**

[Union[Face, list[Face]]] The face or faces to replace.

replacement_selection

[Union[Face, list[Face]]] The face or faces to replace with.

Returns

bool

True when successful, False when failed.

GeometryCommands.**split_body**(*bodies*: list[ansys.geometry.core.designer.body.Body], *plane*: ansys.geometry.core.math.plane.Plane, *slicers*: ansys.geometry.core.designer.edge.Edge | list[ansys.geometry.core.designer.edge.Edge] | ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *faces*: list[ansys.geometry.core.designer.face.Face], *extendfaces*: bool) → bool

Split bodies with a plane, slicers, or faces.

Parameters**bodies**

[list[Body]] Bodies to split.

plane

[Plane] Plane to split with

slicers

[Edge | list[Edge] | Face | list[Face]] Slicers to split with.

faces

[list[Face]] Faces to split with.

extendFaces

[bool] Extend faces if split with faces.

Returns**bool**

True when successful, False when failed.

GeometryCommands.**get_round_info**(*face*: ansys.geometry.core.designer.face.Face) → tuple[bool, ansys.geometry.core.typing.Real]

Get info on the rounding of a face.

Parameters**Face**

The design face to get round info on.

Returns**tuple[bool, Real]**

True if round is aligned with face's U-parameter direction, False otherwise. Radius of the round.

ExtrudeType

class ansys.geometry.core.designer.geometry_commands.**ExtrudeType**(*args, **kwargs)

Bases: `enum.Enum`

Provides values for extrusion types.

Overview

Attributes

<i>NONE</i>
<i>ADD</i>
<i>CUT</i>
<i>FORCE_ADD</i>
<i>FORCE_CUT</i>
<i>FORCE_INDEPENDENT</i>
<i>FORCE_NEW_SURFACE</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import ExtrudeType
```

Attribute detail

ExtrudeType.**NONE** = 0

ExtrudeType.**ADD** = 1

ExtrudeType.**CUT** = 2

ExtrudeType.**FORCE_ADD** = 3

ExtrudeType.**FORCE_CUT** = 4

ExtrudeType.**FORCE_INDEPENDENT** = 5

ExtrudeType.**FORCE_NEW_SURFACE** = 6

OffsetMode

```
class ansys.geometry.core.designer.geometry_commands.OffsetMode(*args, **kws)
```

Bases: enum.Enum

Provides values for offset modes during extrusions.

Overview

Attributes

<i>IGNORE_RELATIONSHIPS</i>
<i>MOVE_FACES_TOGETHER</i>
<i>MOVE_FACES_APART</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import OffsetMode
```

Attribute detail

OffsetMode.IGNORE_RELATIONSHIPS = 0

OffsetMode.MOVE_FACES_TOGETHER = 1

OffsetMode.MOVE_FACES_APART = 2

FillPatternType

class ansys.geometry.core.designer.geometry_commands.FillPatternType(*args, **kws)

Bases: `enum.Enum`

Provides values for types of fill patterns.

Overview

Attributes

<i>GRID</i>
<i>OFFSET</i>
<i>SKEWED</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import FillPatternType
```

Attribute detail

FillPatternType.GRID = 0

FillPatternType.OFFSET = 1

FillPatternType.SKEWED = 2

Description

Provides tools for pulling geometry.

The part.py module

Summary

Classes

<i>Part</i>	Represents a part master.
<i>MasterComponent</i>	Represents a part occurrence.

Part

```
class ansys.geometry.core.designer.part.Part(id: str, name: str, components: list[MasterComponent],
                                             bodies:
                                             list[ansys.geometry.core.designer.body.MasterBody])
```

Represents a part master.

This class should not be accessed by users. The Part class holds fundamental data of an assembly.

Parameters

id

[str] Unique identifier for the part.

name

[str] Name of the part.

components

[list[MasterComponent]] list of MasterComponent children that the part contains.

bodies

[list[MasterBody]] list of MasterBody children that the part contains. These are master bodies.

Overview

Properties

<i>id</i>	ID of the part.
<i>name</i>	Name of the part.
<i>components</i>	MasterComponent children that the part contains.
<i>bodies</i>	MasterBody children that the part contains.

Special methods

<code>__repr__</code>	Represent the part as a string.
-----------------------	---------------------------------

Import detail

```
from ansys.geometry.core.designer.part import Part
```

Property detail

property Part.**id**: str

ID of the part.

property Part.**name**: str

Name of the part.

property Part.**components**: list[MasterComponent]

MasterComponent children that the part contains.

property Part.**bodies**: list[ansys.geometry.core.designer.body.MasterBody]

MasterBody children that the part contains.

These are master bodies.

Method detail

Part.__repr__() → str

Represent the part as a string.

MasterComponent

```
class ansys.geometry.core.designer.part.MasterComponent(id: str, name: str, part: Part, transform:
ansys.geometry.core.math.matrix.Matrix44
= IDENTITY_MATRIX44)
```

Represents a part occurrence.

Parameters

id

[str] Unique identifier for the transformed part.

name

[str] Name of the transformed part.

part

[Part] Reference to the transformed part's master part.

transform

[Matrix44] 4x4 transformation matrix from the master part.

Notes

This class should not be accessed by users. It holds the fundamental data of an assembly. Master components wrap parts by adding a transform matrix.

Overview

Properties

<i>id</i>	ID of the transformed part.
<i>name</i>	Name of the transformed part.
<i>occurrences</i>	List of all occurrences of the component.
<i>part</i>	Master part of the transformed part.
<i>transform</i>	4x4 transformation matrix from the master part.

Special methods

<code>__repr__</code>	Represent the master component as a string.
-----------------------	---

Import detail

```
from ansys.geometry.core.designer.part import MasterComponent
```

Property detail

property MasterComponent.id: `str`

ID of the transformed part.

property MasterComponent.name: `str`

Name of the transformed part.

property MasterComponent.occurrences:

`list[ansys.geometry.core.designer.component.Component]`

List of all occurrences of the component.

property MasterComponent.part: `Part`

Master part of the transformed part.

property MasterComponent.transform: `ansys.geometry.core.math.matrix.Matrix44`

4x4 transformation matrix from the master part.

Method detail

MasterComponent.__repr__() → `str`

Represent the master component as a string.

Description

Module providing fundamental data of an assembly.

The selection.py module

Summary

Classes

<code>NamedSelection</code>	Represents a single named selection within the design assembly.
-----------------------------	---

NamedSelection

```
class ansys.geometry.core.designer.selection.NamedSelection(name: str, grpc_client: ansys.geometry.core.connection.client.GrpcClient,
    bodies: list[ansys.geometry.core.designer.body.Body] | None = None, faces: list[ansys.geometry.core.designer.face.Face] | None = None, edges: list[ansys.geometry.core.designer.edge.Edge] | None = None, beams: list[ansys.geometry.core.designer.beam.Beam] | None = None, design_points: list[ansys.geometry.core.designer.designpoint.DesignPoint] | None = None, preexisting_id: str | None = None)
```

Represents a single named selection within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

A named selection organizes one or more design entities together for common actions against the entire group.

Parameters

name

[**str**] User-defined name for the named selection.

grpc_client

[**GrpcClient**] Active supporting Geometry service instance for design modeling.

bodies

[**list**[**Body**], default: **None**] All bodies to include in the named selection.

faces

[**list**[**Face**], default: **None**] All faces to include in the named selection.

edges

[**list**[**Edge**], default: **None**] All edges to include in the named selection.

beams

[**list**[**Beam**], default: **None**] All beams to include in the named selection.

design_points

[**list**[**DesignPoints**], default: **None**] All design points to include in the named selection.

Overview

Properties

<i>id</i>	ID of the named selection.
<i>name</i>	Name of the named selection.

Import detail

```
from ansys.geometry.core.designer.selection import NamedSelection
```

Property detail

property `NamedSelection.id: str`

ID of the named selection.

property `NamedSelection.name: str`

Name of the named selection.

Description

Module for creating a named selection.

Description

PyAnsys Geometry designer subpackage.

The materials package

Summary

Submodules

<i>material</i>	Provides the data structure for material and material properties.
<i>property</i>	Provides the MaterialProperty class.

The material.py module

Summary

Classes

<i>Material</i>	Provides the data structure for a material.
-----------------	---

Material

class ansys.geometry.core.materials.material.**Material**(*name: str, density: pint.Quantity, additional_properties: collections.abc.Sequence[ansys.geometry.core.materials.property.MaterialProperty] | None = None*)

Provides the data structure for a material.

Parameters

name: str

Material name.

density: ~pint.Quantity

Material density.

additional_properties: Sequence[MaterialProperty], default: None

Additional material properties.

Overview

Methods

<i>add_property</i>	Add a material property to the Material class.
---------------------	--

Properties

<i>properties</i>	Dictionary of the material property type and material properties.
<i>name</i>	Material name.

Import detail

```
from ansys.geometry.core.materials.material import Material
```

Property detail

property `Material.properties:`
dict[`ansys.geometry.core.materials.property.MaterialPropertyType`,
`ansys.geometry.core.materials.property.MaterialProperty`]

Dictionary of the material property type and material properties.

property `Material.name:` **str**

Material name.

Method detail

`Material.add_property`(*type*: `ansys.geometry.core.materials.property.MaterialPropertyType`, *name*: *str*,
quantity: `pint.Quantity`) → `None`

Add a material property to the `Material` class.

Parameters

type
`[MaterialPropertyType]` Material property type.

name: **str**
 Material name.

quantity: `~pint.Quantity`
 Material value and unit.

Description

Provides the data structure for material and material properties.

The `property.py` module

Summary

Classes

<code>MaterialProperty</code>	Provides the data structure for a material property.
-------------------------------	--

Enums

<code>MaterialPropertyType</code>	Enum holding the possible values for <code>MaterialProperty</code> objects.
-----------------------------------	---

`MaterialProperty`

class `ansys.geometry.core.materials.property.MaterialProperty`(*type*: `MaterialPropertyType` | *str*,
name: *str*, *quantity*: `pint.Quantity` |
`ansys.geometry.core.typing.Real`)

Provides the data structure for a material property.

Parameters

type
`[MaterialPropertyType` | `str`] Type of the material property. If the type is a string, it

must be a valid material property type - though it might not be supported by the MaterialPropertyType enum.

name: str

Material property name.

quantity: ~pint.Quantity | Real

Value and unit in case of a supported Quantity. If the type is not supported, it must be a Real value (float or integer).

Overview

Properties

<i>type</i>	Material property ID.
<i>name</i>	Material property name.
<i>quantity</i>	Material property quantity and unit.

Import detail

```
from ansys.geometry.core.materials.property import MaterialProperty
```

Property detail

property MaterialProperty.**type**: *MaterialPropertyType* | **str**

Material property ID.

If the type is not supported, it will be a string.

property MaterialProperty.**name**: **str**

Material property name.

property MaterialProperty.**quantity**: *pint.Quantity* | *ansys.geometry.core.typing.Real*

Material property quantity and unit.

If the type is not supported, it will be a Real value (float or integer).

MaterialPropertyType

class *ansys.geometry.core.materials.property.MaterialPropertyType*(*args, **kws)

Bases: *enum.Enum*

Enum holding the possible values for MaterialProperty objects.

Overview

Attributes

<i>DENSITY</i>
<i>ELASTIC_MODULUS</i>
<i>POISSON_RATIO</i>
<i>SHEAR_MODULUS</i>
<i>SPECIFIC_HEAT</i>
<i>TENSILE_STRENGTH</i>
<i>THERMAL_CONDUCTIVITY</i>

Static methods

<code>from_id</code>	Return the MaterialPropertyType value from the service.
----------------------	---

Import detail

```
from ansys.geometry.core.materials.property import MaterialPropertyType
```

Attribute detail

MaterialPropertyType.DENSITY = 'Density'

MaterialPropertyType.ELASTIC_MODULUS = 'ElasticModulus'

MaterialPropertyType.POISSON_RATIO = 'PoissonsRatio'

MaterialPropertyType.SHEAR_MODULUS = 'ShearModulus'

MaterialPropertyType.SPECIFIC_HEAT = 'SpecificHeat'

MaterialPropertyType.TENSILE_STRENGTH = 'TensileStrength'

MaterialPropertyType.THERMAL_CONDUCTIVITY = 'ThermalConductivity'

Method detail

static MaterialPropertyType.**from_id**(*id: str*) → *MaterialPropertyType*

Return the MaterialPropertyType value from the service.

Parameters

id

[str] Geometry Service string representation of a property type.

Returns

MaterialPropertyType

Common name for property type.

Description

Provides the MaterialProperty class.

Description

PyAnsys Geometry materials subpackage.

The math package

Summary

Submodules

<i>bbox</i>	Provides for managing a bounding box.
<i>constants</i>	Provides mathematical constants.
<i>frame</i>	Provides for managing a frame.
<i>matrix</i>	Provides matrix primitive representations.
<i>misc</i>	Provides auxiliary math functions for PyAnsys Geometry.
<i>plane</i>	Provides primitive representation of a 2D plane in 3D space.
<i>point</i>	Provides geometry primitive representation for 2D and 3D points.
<i>vector</i>	Provides for creating and managing 2D and 3D vectors.

The `bbox.py` module

Summary

Classes

<i>BoundingBox2D</i>	Maintains the X and Y dimensions.
----------------------	-----------------------------------

BoundingBox2D

```
class ansys.geometry.core.math.bbox.BoundingBox2D(x_min: ansys.geometry.core.typing.Real =
    sys.float_info.max, x_max:
    ansys.geometry.core.typing.Real =
    sys.float_info.min, y_min:
    ansys.geometry.core.typing.Real =
    sys.float_info.max, y_max:
    ansys.geometry.core.typing.Real =
    sys.float_info.min)
```

Maintains the X and Y dimensions.

Parameters

- x_min**
[Real] Minimum value for the x-dimensional bounds.
- x_max**
[Real] Maximum value for the x-dimensional bounds.
- y_min**
[Real] Minimum value for the y-dimensional bounds.
- y_max**
[Real] Maximum value for the y-dimensional bounds.

Overview

Methods

<i>add_point</i>	Extend the ranges of the bounding box to include a point.
<i>add_point_components</i>	Extend the ranges of the bounding box to include the X and Y values.
<i>add_points</i>	Extend the ranges of the bounding box to include given points.
<i>contains_point</i>	Evaluate whether a point lies within the X and Y range bounds.
<i>contains_point_components</i>	Check if point components are within the X and Y range bounds.

Properties

<code>x_min</code>	Minimum value of X-dimensional bounds.
<code>x_max</code>	Maximum value of the X-dimensional bounds.
<code>y_min</code>	Minimum value of Y-dimensional bounds.
<code>y_max</code>	Maximum value of Y-dimensional bounds.

Static methods

<code>intersect_bboxes</code>	Find the intersection of 2 BoundingBox2D objects.
-------------------------------	---

Special methods

<code>__eq__</code>	Equals operator for the BoundingBox2D class.
<code>__ne__</code>	Not equals operator for the BoundingBox2D class.

Import detail

```
from ansys.geometry.core.math.bbox import BoundingBox2D
```

Property detail

property BoundingBox2D.`x_min`: `ansys.geometry.core.typing.Real`
Minimum value of X-dimensional bounds.

Returns

Real

Minimum value of the X-dimensional bounds.

property BoundingBox2D.`x_max`: `ansys.geometry.core.typing.Real`
Maximum value of the X-dimensional bounds.

Returns

Real

Maximum value of the X-dimensional bounds.

property BoundingBox2D.`y_min`: `ansys.geometry.core.typing.Real`
Minimum value of Y-dimensional bounds.

Returns

Real

Minimum value of Y-dimensional bounds.

property BoundingBox2D.`y_max`: `ansys.geometry.core.typing.Real`
Maximum value of Y-dimensional bounds.

Returns

Real

Maximum value of Y-dimensional bounds.

Method detail

`BoundingBox2D.add_point` (*point*: `ansys.geometry.core.math.point.Point2D`) → `None`

Extend the ranges of the bounding box to include a point.

Parameters

point

[`Point2D`] Point to include within the bounds.

Notes

This method is only applicable if the point components are outside the current bounds.

`BoundingBox2D.add_point_components` (*x*: `ansys.geometry.core.typing.Real`, *y*: `ansys.geometry.core.typing.Real`) → `None`

Extend the ranges of the bounding box to include the X and Y values.

Parameters

x

[`Real`] Point X component to include within the bounds.

y

[`Real`] Point Y component to include within the bounds.

Notes

This method is only applicable if the point components are outside the current bounds.

`BoundingBox2D.add_points` (*points*: `list[ansys.geometry.core.math.point.Point2D]`) → `None`

Extend the ranges of the bounding box to include given points.

Parameters

points

[`list[Point2D]`] List of points to include within the bounds.

`BoundingBox2D.contains_point` (*point*: `ansys.geometry.core.math.point.Point2D`) → `bool`

Evaluate whether a point lies within the X and Y range bounds.

Parameters

point

[`Point2D`] Point to compare against the bounds.

Returns

bool

True if the point is contained in the bounding box. Otherwise, False.

`BoundingBox2D.contains_point_components` (*x*: `ansys.geometry.core.typing.Real`, *y*: `ansys.geometry.core.typing.Real`) → `bool`

Check if point components are within the X and Y range bounds.

Parameters

x

[`Real`] Point X component to compare against the bounds.

y

[`Real`] Point Y component to compare against the bounds.

Returns**bool**

True if the components are contained in the bounding box. Otherwise, False.

`BoundingBox2D.__eq__(other: BoundingBox2D) → bool`

Equals operator for the `BoundingBox2D` class.

`BoundingBox2D.__ne__(other: BoundingBox2D) → bool`

Not equals operator for the `BoundingBox2D` class.

static `BoundingBox2D.intersect_bboxes(box_1: BoundingBox2D, box_2: BoundingBox2D) → None | BoundingBox2D`

Find the intersection of 2 `BoundingBox2D` objects.

Parameters**box_1: BoundingBox2D**

The box to consider the intersection of with respect to `box_2`.

box_2: BoundingBox2D

The box to consider the intersection of with respect to `box_1`.

Returns**BoundingBox2D:**

The box representing the intersection of the two passed in boxes.

Description

Provides for managing a bounding box.

The constants.py module**Summary****Constants**

<code>DEFAULT_POINT3D</code>	Default value for a 3D point.
<code>DEFAULT_POINT2D</code>	Default value for a 2D point.
<code>IDENTITY_MATRIX33</code>	Identity for a <code>Matrix33</code> object.
<code>IDENTITY_MATRIX44</code>	Identity for a <code>Matrix44</code> object.
<code>UNITVECTOR3D_X</code>	Default 3D unit vector in the Cartesian traditional X direction.
<code>UNITVECTOR3D_Y</code>	Default 3D unit vector in the Cartesian traditional Y direction.
<code>UNITVECTOR3D_Z</code>	Default 3D unit vector in the Cartesian traditional Z direction.
<code>UNITVECTOR2D_X</code>	Default 2D unit vector in the Cartesian traditional X direction.
<code>UNITVECTOR2D_Y</code>	Default 2D unit vector in the Cartesian traditional Y direction.
<code>ZERO_VECTOR3D</code>	Zero-valued <code>Vector3D</code> object.
<code>ZERO_VECTOR2D</code>	Zero-valued <code>Vector2D</code> object.
<code>ZERO_POINT3D</code>	Zero-valued <code>Point3D</code> object.
<code>ZERO_POINT2D</code>	Zero-valued <code>Point2D</code> object.

Description

Provides mathematical constants.

Module detail

constants.**DEFAULT_POINT3D**

Default value for a 3D point.

constants.**DEFAULT_POINT2D**

Default value for a 2D point.

constants.**IDENTITY_MATRIX33**

Identity for a Matrix33 object.

constants.**IDENTITY_MATRIX44**

Identity for a Matrix44 object.

constants.**UNITVECTOR3D_X**

Default 3D unit vector in the Cartesian traditional X direction.

constants.**UNITVECTOR3D_Y**

Default 3D unit vector in the Cartesian traditional Y direction.

constants.**UNITVECTOR3D_Z**

Default 3D unit vector in the Cartesian traditional Z direction.

constants.**UNITVECTOR2D_X**

Default 2D unit vector in the Cartesian traditional X direction.

constants.**UNITVECTOR2D_Y**

Default 2D unit vector in the Cartesian traditional Y direction.

constants.**ZERO_VECTOR3D**

Zero-valued Vector3D object.

constants.**ZERO_VECTOR2D**

Zero-valued Vector2D object.

constants.**ZERO_POINT3D**

Zero-valued Point3D object.

constants.**ZERO_POINT2D**

Zero-valued Point2D object.

The `frame.py` module

Summary

Classes

<i>Frame</i>	Representation of a frame.
--------------	----------------------------

Frame

```
class ansys.geometry.core.math.frame.Frame(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D =
    ZERO_POINT3D, direction_x: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_X, direction_y: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_Y)
```

Representation of a frame.

Parameters

origin

[[ndarray](#) | [RealSequence](#) | [Point3D](#), default: `ZERO_POINT3D`] Centered origin of the frame. The default is `ZERO_POINT3D`, which is the Cartesian origin.

direction_x

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#), default: `UNITVECTOR3D_X`] X-axis direction.

direction_y

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#), default: `UNITVECTOR3D_Y`] Y-axis direction.

Overview

Methods

<code>transform_point2d_local_to_global</code>	Transform a 2D point to a global 3D point.
--	--

Properties

<code>origin</code>	Origin of the frame.
<code>direction_x</code>	X-axis direction of the frame.
<code>direction_y</code>	Y-axis direction of the frame.
<code>direction_z</code>	Z-axis direction of the frame.
<code>global_to_local_rotation</code>	Global to local space transformation matrix.
<code>local_to_global_rotation</code>	Local to global space transformation matrix.
<code>transformation_matrix</code>	Full 4x4 transformation matrix.

Special methods

<code>__eq__</code>	Equals operator for the Frame class.
<code>__ne__</code>	Not equals operator for the Frame class.

Import detail

```
from ansys.geometry.core.math.frame import Frame
```

Property detail

property `Frame.origin`: *ansys.geometry.core.math.point.Point3D*

Origin of the frame.

property `Frame.direction_x`: *ansys.geometry.core.math.vector.UnitVector3D*

X-axis direction of the frame.

property `Frame.direction_y`: *ansys.geometry.core.math.vector.UnitVector3D*

Y-axis direction of the frame.

property `Frame.direction_z`: *ansys.geometry.core.math.vector.UnitVector3D*

Z-axis direction of the frame.

property `Frame.global_to_local_rotation`: *ansys.geometry.core.math.matrix.Matrix33*

Global to local space transformation matrix.

Returns

Matrix33

3x3 matrix representing the transformation from global to local coordinate space, excluding origin translation.

property `Frame.local_to_global_rotation`: *ansys.geometry.core.math.matrix.Matrix33*

Local to global space transformation matrix.

Returns

Matrix33

3x3 matrix representing the transformation from local to global coordinate space.

property `Frame.transformation_matrix`: *ansys.geometry.core.math.matrix.Matrix44*

Full 4x4 transformation matrix.

Returns

Matrix44

4x4 matrix representing the transformation from global to local coordinate space.

Method detail

`Frame.transform_point2d_local_to_global`(*point*: *ansys.geometry.core.math.point.Point2D*) → *ansys.geometry.core.math.point.Point3D*

Transform a 2D point to a global 3D point.

This method transforms a local, plane-contained `Point2D` object in the global coordinate system, thus representing it as a `Point3D` object.

Parameters

point

[`Point2D`] `Point2D` local object to express in global coordinates.

Returns

Point3D

Global coordinates for the 3D point.

`Frame.__eq__(other: Frame) → bool`

Equals operator for the Frame class.

`Frame.__ne__(other: Frame) → bool`

Not equals operator for the Frame class.

Description

Provides for managing a frame.

The matrix.py module**Summary****Classes**

<i>Matrix</i>	Provides matrix representation.
<i>Matrix33</i>	Provides 3x3 matrix representation.
<i>Matrix44</i>	Provides 4x4 matrix representation.

Constants

<i>DEFAULT_MATRIX33</i>	Default value of the 3x3 identity matrix for the <i>Matrix33</i> class.
<i>DEFAULT_MATRIX44</i>	Default value of the 4x4 identity matrix for the <i>Matrix44</i> class.

Matrix

`class ansys.geometry.core.math.matrix.Matrix(shape, dtype=float, buffer=None, offset=0, strides=None, order=None)`

Bases: `numpy.ndarray`

Provides matrix representation.

Parameters**input**

[`ndarray` | `RealSequence`] Matrix arguments as a `np.ndarray` class.

Overview**Methods**

<i>determinant</i>	Get the determinant of the matrix.
<i>inverse</i>	Provide the inverse of the matrix.

Special methods

<code>__mul__</code>	Get the multiplication of the matrix.
<code>__eq__</code>	Equals operator for the <code>Matrix</code> class.
<code>__ne__</code>	Not equals operator for the <code>Matrix</code> class.

Import detail

```
from ansys.geometry.core.math.matrix import Matrix
```

Method detail

`Matrix.determinant()` → `ansys.geometry.core.typing.Real`

Get the determinant of the matrix.

`Matrix.inverse()` → `Matrix`

Provide the inverse of the matrix.

`Matrix.__mul__(other: Matrix | numpy.ndarray)` → `Matrix`

Get the multiplication of the matrix.

`Matrix.__eq__(other: Matrix)` → `bool`

Equals operator for the `Matrix` class.

`Matrix.__ne__(other: Matrix)` → `bool`

Not equals operator for the `Matrix` class.

Matrix33

```
class ansys.geometry.core.math.matrix.Matrix33(shape, dtype=float, buffer=None, offset=0,
                                               strides=None, order=None)
```

Bases: `Matrix`

Provides 3x3 matrix representation.

Parameters

input

[`ndarray` | `RealSequence` | `Matrix`, default: `DEFAULT_MATRIX33`] Matrix arguments as a `np.ndarray` class.

Import detail

```
from ansys.geometry.core.math.matrix import Matrix33
```

Matrix44

```
class ansys.geometry.core.math.matrix.Matrix44(shape, dtype=float, buffer=None, offset=0,
                                               strides=None, order=None)
```

Bases: `Matrix`

Provides 4x4 matrix representation.

Parameters

input

[`ndarray` | `RealSequence` | `Matrix`, default: `DEFAULT_MATRIX44`] Matrix arguments as a `np.ndarray` class.

Overview**Constructors**

<code>create_translation</code>	Create a matrix representing the specified translation.
<code>create_rotation</code>	Create a matrix representing the specified rotation.

Methods

<code>is_translation</code>	Check if the matrix represents a translation.
-----------------------------	---

Import detail

```
from ansys.geometry.core.math.matrix import Matrix44
```

Method detail

classmethod `Matrix44.create_translation`(*translation*: `ansys.geometry.core.math.vector.Vector3D`) → `Matrix44`

Create a matrix representing the specified translation.

Parameters**translation**

[`Vector3D`] The translation vector representing the translation. The components of the vector should be in meters.

Returns**Matrix44**

A 4x4 matrix representing the translation.

Examples

```
>>> translation_vector = Vector3D(1.0, 2.0, 3.0)
>>> translation_matrix = Matrix44.create_translation(translation_vector)
>>> print(translation_matrix)
[[1. 0. 0. 1.]
 [0. 1. 0. 2.]
 [0. 0. 1. 3.]
 [0. 0. 0. 1.]
```

Matrix44.is_translation(*including_identity*: `bool = False`) → `bool`

Check if the matrix represents a translation.

This method checks if the matrix represents a translation transformation. A translation matrix has the following form:

$$\begin{bmatrix} 1 & 0 & 0 & t_x \\ 0 & 1 & 0 & t_y \\ 0 & 0 & 1 & t_z \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

Parameters**including_identity**

[bool, optional] If True, the method will return True for the identity matrix as well. If False, the method will return False for the identity matrix.

Returns**bool**

True if the matrix represents a translation, False otherwise.

Examples

```
>>> matrix = Matrix44([[1, 0, 0, 5], [0, 1, 0, 3], [0, 0, 1, 2], [0, 0, 0, 1]])
>>> matrix.is_translation()
True
>>> identity_matrix = Matrix44([[1, 0, 0, 0], [0, 1, 0, 0], [0, 0, 1, 0], [0, 0, 0, 1]])
>>> identity_matrix.is_translation()
False
>>> identity_matrix.is_translation(including_identity=True)
True
```

classmethod `Matrix44.create_rotation`(*direction_x*: ansys.geometry.core.math.vector.Vector3D, *direction_y*: ansys.geometry.core.math.vector.Vector3D, *direction_z*: ansys.geometry.core.math.vector.Vector3D = None) → *Matrix44*

Create a matrix representing the specified rotation.

Parameters**direction_x**

[Vector3D] The X direction vector.

direction_y

[Vector3D] The Y direction vector.

direction_z

[Vector3D, optional] The Z direction vector. If not provided, it will be calculated as the cross product of *direction_x* and *direction_y*.

Returns**Matrix44**

A 4x4 matrix representing the rotation.

Examples

```
>>> direction_x = Vector3D(1.0, 0.0, 0.0)
>>> direction_y = Vector3D(0.0, 1.0, 0.0)
>>> rotation_matrix = Matrix44.create_rotation(direction_x, direction_y)
>>> print(rotation_matrix)
[[1. 0. 0. 0.]
 [0. 1. 0. 0.]
 [0. 0. 1. 0.]
 [0. 0. 0. 1.]]
```

Description

Provides matrix primitive representations.

Module detail

`matrix.DEFAULT_MATRIX33`

Default value of the 3x3 identity matrix for the `Matrix33` class.

`matrix.DEFAULT_MATRIX44`

Default value of the 4x4 identity matrix for the `Matrix44` class.

The `misc.py` module

Summary

Functions

<code>get_two_circle_intersections</code>	Get the intersection points of two circles.
<code>intersect_interval</code>	Find the intersection of two intervals.

Description

Provides auxiliary math functions for PyAnsys Geometry.

Module detail

`misc.get_two_circle_intersections` (*x0*: `ansys.geometry.core.typing.Real`, *y0*: `ansys.geometry.core.typing.Real`, *r0*: `ansys.geometry.core.typing.Real`, *x1*: `ansys.geometry.core.typing.Real`, *y1*: `ansys.geometry.core.typing.Real`, *r1*: `ansys.geometry.core.typing.Real`)
 → `tuple[tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real], tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real]] | None`

Get the intersection points of two circles.

Parameters

- x0**
[Real] x coordinate of the first circle.
- y0**
[Real] y coordinate of the first circle.
- r0**
[Real] Radius of the first circle.
- x1**
[Real] x coordinate of the second circle.
- y1**
[Real] y coordinate of the second circle.
- r1**
[Real] Radius of the second circle.

Returns**tuple[tuple[Real, Real], tuple[Real, Real]] | None**

Intersection points of the two circles if they intersect. The points are returned as ((x3, y3), (x4, y4)), where (x3, y3) and (x4, y4) are the intersection points of the two circles. If the circles do not intersect, then None is returned.

Notes

This function is based on the following StackOverflow post: <https://stackoverflow.com/questions/55816902/finding-the-intersection-of-two-circles>

That post is based on the following implementation: <https://paulbourke.net/geometry/circlesphere/>

`misc.intersect_interval(first_min, second_min, first_max, second_max) → tuple[bool, ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real]`

Find the intersection of two intervals.

Parameters**first_min**

[Real] The minimum value of the first interval.

second_min

[Real] The minimum value of the second interval.

first_max

[Real] The maximum value of the first interval.

second_max

[Real] The maximum value of the second interval.

Returns**tuple[bool, Real, Real]**

Tuple with a boolean to indicate whether the intervals intersect, the minimum value of the intersection interval, and the maximum value of the intersection interval. If they do not intersect, then the boolean is False and the other values are 0.

The plane.py module**Summary****Classes**

<i>Plane</i>	Provides primitive representation of a 2D plane in 3D space.
--------------	--

Plane

```
class ansys.geometry.core.math.plane.Plane(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D =
    ZERO_POINT3D, direction_x: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_X, direction_y: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_Y)
```

Bases: `ansys.geometry.core.math.frame.Frame`

Provides primitive representation of a 2D plane in 3D space.

Parameters

origin

[`ndarray` | `RealSequence` | `Point3D`, default: `ZERO_POINT3D`] Centered origin of the frame. The default is `ZERO_POINT3D`, which is the Cartesian origin.

direction_x

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`, default: `UNITVECTOR3D_X`] X-axis direction.

direction_y

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`, default: `UNITVECTOR3D_Y`] Y-axis direction.

Overview

Methods

<code>is_point_contained</code>	Check if a 3D point is contained in the plane.
---------------------------------	--

Properties

<code>normal</code>	Calculate the normal vector of the plane.
---------------------	---

Special methods

<code>__eq__</code>	Equals operator for the <code>Plane</code> class.
<code>__ne__</code>	Not equals operator for the <code>Plane</code> class.

Import detail

```
from ansys.geometry.core.math.plane import Plane
```

Property detail

property `Plane.normal`: *ansys.geometry.core.math.vector.UnitVector3D*

Calculate the normal vector of the plane.

Returns

UnitVector3D

Normal vector of the plane.

Method detail

`Plane.is_point_contained`(*point*: *ansys.geometry.core.math.point.Point3D*) → *bool*

Check if a 3D point is contained in the plane.

Parameters

point

[*Point3D*] *Point3D* class to check.

Returns

bool

True if the 3D point is contained in the plane, False otherwise.

`Plane.__eq__`(*other*: *Plane*) → *bool*

Equals operator for the *Plane* class.

`Plane.__ne__`(*other*: *Plane*) → *bool*

Not equals operator for the *Plane* class.

Description

Provides primitive representation of a 2D plane in 3D space.

The `point.py` module

Summary

Classes

<i>Point2D</i>	Provides geometry primitive representation for a 2D point.
<i>Point3D</i>	Provides geometry primitive representation for a 3D point.

Constants

<i>DEFAULT_POINT2D_VALUES</i>	Default values for a 2D point.
<i>DEFAULT_POINT3D_VALUES</i>	Default values for a 3D point.
<i>BASE_UNIT_LENGTH</i>	Default value for the length of the base unit.

Point2D

```
class ansys.geometry.core.math.point.Point2D(input: numpy.ndarray |
                                             ansys.geometry.core.typing.RealSequence =
                                             DEFAULT_POINT2D_VALUES, unit: pint.Unit | None =
                                             None)
```

Bases: `numpy.ndarray`, `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides geometry primitive representation for a 2D point.

Parameters

input

[`ndarray` | `RealSequence`, default: `DEFAULT_POINT2D_VALUES`] Direction arguments, either as a `numpy.ndarray` class or as a `RealSequence`.

unit

[`Unit` | `None`, default: `DEFAULT_UNITS.LENGTH`] Units for defining 2D point values. If not specified, the default unit is `DEFAULT_UNITS.LENGTH`.

Overview

Methods

<code>unit</code>	Get the unit of the object.
<code>base_unit</code>	Get the base unit of the object.

Properties

<code>x</code>	X plane component value.
<code>y</code>	Y plane component value.

Attributes

`flat`

Special methods

<code>__eq__</code>	Equals operator for the <code>Point2D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Point2D</code> class.
<code>__add__</code>	Add operation for the <code>Point2D</code> class.
<code>__sub__</code>	Subtraction operation for the <code>Point2D</code> class.

Import detail

```
from ansys.geometry.core.math.point import Point2D
```

Property detail

property `Point2D.x`: `pint.Quantity`

X plane component value.

property `Point2D.y`: `pint.Quantity`

Y plane component value.

Attribute detail

Point2D.**flat**

Method detail

Point2D.**__eq__**(*other*: Point2D) → bool

Equals operator for the Point2D class.

Point2D.**__ne__**(*other*: Point2D) → bool

Not equals operator for the Point2D class.

Point2D.**__add__**(*other*: Point2D | ansys.geometry.core.math.vector.Vector2D) → Point2D

Add operation for the Point2D class.

Point2D.**__sub__**(*other*: Point2D) → Point2D

Subtraction operation for the Point2D class.

Point2D.**unit**() → pint.Unit

Get the unit of the object.

Point2D.**base_unit**() → pint.Unit

Get the base unit of the object.

Point3D

```
class ansys.geometry.core.math.point.Point3D(input: numpy.ndarray |
                                             ansys.geometry.core.typing.RealSequence =
                                             DEFAULT_POINT3D_VALUES, unit: pint.Unit | None =
                                             None)
```

Bases: `numpy.ndarray`, `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides geometry primitive representation for a 3D point.

Parameters

input

[`ndarray` | `RealSequence`, default: `DEFAULT_POINT3D_VALUES`] Direction arguments, either as a `numpy.ndarray` class or as a `RealSequence`.

unit

[`Unit` | `None`, default: `DEFAULT_UNITS.LENGTH`] Units for defining 3D point values. If not specified, the default unit is `DEFAULT_UNITS.LENGTH`.

Overview

Methods

<code>unit</code>	Get the unit of the object.
<code>base_unit</code>	Get the base unit of the object.
<code>transform</code>	Transform the 3D point with a transformation matrix.

Properties

<code>x</code>	X plane component value.
<code>y</code>	Y plane component value.
<code>z</code>	Z plane component value.

Attributes

`flat`

Special methods

<code>__eq__</code>	Equals operator for the <code>Point3D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Point3D</code> class.
<code>__add__</code>	Add operation for the <code>Point3D</code> class.
<code>__sub__</code>	Subtraction operation for the <code>Point3D</code> class.

Import detail

```
from ansys.geometry.core.math.point import Point3D
```

Property detail

property `Point3D.x`: `pint.Quantity`

X plane component value.

property `Point3D.y`: `pint.Quantity`

Y plane component value.

property `Point3D.z`: `pint.Quantity`

Z plane component value.

Attribute detail

`Point3D.flat`

Method detail

`Point3D.__eq__(other: Point3D) → bool`

Equals operator for the `Point3D` class.

`Point3D.__ne__(other: Point3D) → bool`

Not equals operator for the `Point3D` class.

`Point3D.__add__(other: Point3D | ansys.geometry.core.math.vector.Vector3D) → Point3D`

Add operation for the `Point3D` class.

`Point3D.__sub__(other: Point3D | ansys.geometry.core.math.vector.Vector3D) → Point3D`

Subtraction operation for the `Point3D` class.

`Point3D.unit()` → `pint.Unit`

Get the unit of the object.

`Point3D.base_unit()` → `pint.Unit`

Get the base unit of the object.

`Point3D.transform(matrix: ansys.geometry.core.math.matrix.Matrix44)` → `Point3D`

Transform the 3D point with a transformation matrix.

Parameters

matrix

[Matrix44] 4x4 transformation matrix to apply to the point.

Returns

Point3D

New 3D point that is the transformed copy of the original 3D point after applying the transformation matrix.

Notes

Transform the `Point3D` object by applying the specified 4x4 transformation matrix and return a new `Point3D` object representing the transformed point.

Description

Provides geometry primitive representation for 2D and 3D points.

Module detail

`point.DEFAULT_POINT2D_VALUES`

Default values for a 2D point.

`point.DEFAULT_POINT3D_VALUES`

Default values for a 3D point.

`point.BASE_UNIT_LENGTH`

Default value for the length of the base unit.

The `vector.py` module

Summary

Classes

<code>Vector3D</code>	Provides for managing and creating a 3D vector.
<code>Vector2D</code>	Provides for creating and managing a 2D vector.
<code>UnitVector3D</code>	Provides for creating and managing a 3D unit vector.
<code>UnitVector2D</code>	Provides for creating and managing a 3D unit vector.

Vector3D

```
class ansys.geometry.core.math.vector.Vector3D(shape, dtype=float, buffer=None, offset=0,
                                             strides=None, order=None)
```

Bases: `numpy.ndarray`

Provides for managing and creating a 3D vector.

Parameters

input

[`ndarray` | `RealSequence`] 3D `numpy.ndarray` class with shape(X,).

Overview

Constructors

<code>from_points</code>	Create a 3D vector from two distinct 3D points.
--------------------------	---

Methods

<code>is_perpendicular_to</code>	Check if this vector and another vector are perpendicular.
<code>is_parallel_to</code>	Check if this vector and another vector are parallel.
<code>is_opposite</code>	Check if this vector and another vector are opposite.
<code>normalize</code>	Return a normalized version of the 3D vector.
<code>transform</code>	Transform the 3D vector3D with a transformation matrix.
<code>get_angle_between</code>	Get the angle between this 3D vector and another 3D vector.
<code>cross</code>	Return the cross product of Vector3D objects.

Properties

<code>x</code>	X coordinate of the Vector3D class.
<code>y</code>	Y coordinate of the Vector3D class.
<code>z</code>	Z coordinate of the Vector3D class.
<code>norm</code>	Norm of the vector.
<code>magnitude</code>	Norm of the vector.
<code>is_zero</code>	Check if all components of the 3D vector are zero.

Special methods

<code>__eq__</code>	Equals operator for the Vector3D class.
<code>__ne__</code>	Not equals operator for the Vector3D class.
<code>__mul__</code>	Overload * operator with dot product.
<code>__mod__</code>	Overload % operator with cross product.
<code>__add__</code>	Addition operation overload for 3D vectors.
<code>__sub__</code>	Subtraction operation overload for 3D vectors.

Import detail

```
from ansys.geometry.core.math.vector import Vector3D
```


Property detail

property `Vector3D.x`: `ansys.geometry.core.typing.Real`

X coordinate of the `Vector3D` class.

property `Vector3D.y`: `ansys.geometry.core.typing.Real`

Y coordinate of the `Vector3D` class.

property `Vector3D.z`: `ansys.geometry.core.typing.Real`

Z coordinate of the `Vector3D` class.

property `Vector3D.norm`: `float`

Norm of the vector.

property `Vector3D.magnitude`: `float`

Norm of the vector.

property `Vector3D.is_zero`: `bool`

Check if all components of the 3D vector are zero.

Method detail

`Vector3D.is_perpendicular_to`(*other_vector*: `Vector3D`) → `bool`

Check if this vector and another vector are perpendicular.

`Vector3D.is_parallel_to`(*other_vector*: `Vector3D`) → `bool`

Check if this vector and another vector are parallel.

`Vector3D.is_opposite`(*other_vector*: `Vector3D`) → `bool`

Check if this vector and another vector are opposite.

`Vector3D.normalize`() → `Vector3D`

Return a normalized version of the 3D vector.

`Vector3D.transform`(*matrix*: `ansys.geometry.core.math.matrix.Matrix44`) → `Vector3D`

Transform the 3D vector `Vector3D` with a transformation matrix.

Parameters

matrix

[`Matrix44`] 4x4 transformation matrix to apply to the vector.

Returns

Vector3D

A new 3D vector that is the transformed copy of the original 3D vector after applying the transformation matrix.

Notes

Transform the `Vector3D` object by applying the specified 4x4 transformation matrix and return a new `Vector3D` object representing the transformed vector.

`Vector3D.get_angle_between`(*v*: `Vector3D`) → `pint.Quantity`

Get the angle between this 3D vector and another 3D vector.

Parameters

v

[`Vector3D`] Other 3D vector for computing the angle.

Returns**Quantity**

Angle between these two 3D vectors.

`Vector3D.cross(v: Vector3D) → Vector3D`

Return the cross product of `Vector3D` objects.

`Vector3D.__eq__(other: Vector3D) → bool`

Equals operator for the `Vector3D` class.

`Vector3D.__ne__(other: Vector3D) → bool`

Not equals operator for the `Vector3D` class.

`Vector3D.__mul__(other: Vector3D | ansys.geometry.core.typing.Real) → Vector3D | ansys.geometry.core.typing.Real`

Overload `*` operator with dot product.

Notes

This method also admits scalar multiplication.

`Vector3D.__mod__(other: Vector3D) → Vector3D`

Overload `%` operator with cross product.

`Vector3D.__add__(other: Vector3D | ansys.geometry.core.math.point.Point3D) → Vector3D | ansys.geometry.core.math.point.Point3D`

Addition operation overload for 3D vectors.

`Vector3D.__sub__(other: Vector3D) → Vector3D`

Subtraction operation overload for 3D vectors.

classmethod `Vector3D.from_points(point_a: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D, point_b: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D)`

Create a 3D vector from two distinct 3D points.

Parameters**point_a**

`[ndarray | RealSequence | Point3D]` `Point3D` class representing the first point.

point_b

`[ndarray | RealSequence | Point3D]` `Point3D` class representing the second point.

Returns**Vector3D**

3D vector from `point_a` to `point_b`.

Notes

The resulting 3D vector is always expressed in `Point3D` base units.

Vector2D

class `ansys.geometry.core.math.vector.Vector2D`(*shape*, *dtype=float*, *buffer=None*, *offset=0*, *strides=None*, *order=None*)

Bases: `numpy.ndarray`

Provides for creating and managing a 2D vector.

Parameters

input

[`ndarray` | `RealSequence`] 2D `numpy.ndarray` class with shape(X,).

Overview

Constructors

<code>from_points</code>	Create a 2D vector from two distinct 2D points.
--------------------------	---

Methods

<code>cross</code>	Return the cross product of <code>Vector2D</code> objects.
<code>is_perpendicular_to</code>	Check if this 2D vector and another 2D vector are perpendicular.
<code>is_parallel_to</code>	Check if this vector and another vector are parallel.
<code>is_opposite</code>	Check if this vector and another vector are opposite.
<code>normalize</code>	Return a normalized version of the 2D vector.
<code>get_angle_between</code>	Get the angle between this 2D vector and another 2D vector.

Properties

<code>x</code>	X coordinate of the 2D vector.
<code>y</code>	Y coordinate of the 2D vector.
<code>norm</code>	Norm of the 2D vector.
<code>magnitude</code>	Norm of the 2D vector.
<code>is_zero</code>	Check if values for all components of the 2D vector are zero.

Special methods

<code>__eq__</code>	Equals operator for the <code>Vector2D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Vector2D</code> class.
<code>__mul__</code>	Overload <code>*</code> operator with dot product.
<code>__add__</code>	Addition operation overload for 2D vectors.
<code>__sub__</code>	Subtraction operation overload for 2D vectors.
<code>__mod__</code>	Overload <code>%</code> operator with cross product.

Import detail

```
from ansys.geometry.core.math.vector import Vector2D
```

Property detail

property `Vector2D.x`: `ansys.geometry.core.typing.Real`

X coordinate of the 2D vector.

property `Vector2D.y`: `ansys.geometry.core.typing.Real`

Y coordinate of the 2D vector.

property `Vector2D.norm`: `float`

Norm of the 2D vector.

property `Vector2D.magnitude`: `float`

Norm of the 2D vector.

property `Vector2D.is_zero`: `bool`

Check if values for all components of the 2D vector are zero.

Method detail

`Vector2D.cross`(*v*: `Vector2D`) → `ansys.geometry.core.typing.Real`

Return the cross product of `Vector2D` objects.

`Vector2D.is_perpendicular_to`(*other_vector*: `Vector2D`) → `bool`

Check if this 2D vector and another 2D vector are perpendicular.

`Vector2D.is_parallel_to`(*other_vector*: `Vector2D`) → `bool`

Check if this vector and another vector are parallel.

`Vector2D.is_opposite`(*other_vector*: `Vector2D`) → `bool`

Check if this vector and another vector are opposite.

`Vector2D.normalize`() → `Vector2D`

Return a normalized version of the 2D vector.

`Vector2D.get_angle_between`(*v*: `Vector2D`) → `pint.Quantity`

Get the angle between this 2D vector and another 2D vector.

Parameters

v

[`Vector2D`] Other 2D vector to compute the angle with.

Returns

Quantity

Angle between these two 2D vectors.

`Vector2D.__eq__`(*other*: `Vector2D`) → `bool`

Equals operator for the `Vector2D` class.

`Vector2D.__ne__`(*other*: `Vector2D`) → `bool`

Not equals operator for the `Vector2D` class.

`Vector2D.__mul__`(*other*: `Vector2D` | `ansys.geometry.core.typing.Real`) → `Vector2D` | `ansys.geometry.core.typing.Real`

Overload * operator with dot product.

Notes

This method also admits scalar multiplication.

`Vector2D.__add__(other: Vector2D | ansys.geometry.core.math.point.Point2D) → Vector2D | ansys.geometry.core.math.point.Point2D`

Addition operation overload for 2D vectors.

`Vector2D.__sub__(other: Vector2D) → Vector2D`

Subtraction operation overload for 2D vectors.

`Vector2D.__mod__(other: Vector2D) → ansys.geometry.core.typing.Real`

Overload % operator with cross product.

classmethod `Vector2D.from_points(point_a: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point2D, point_b: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point2D)`

Create a 2D vector from two distinct 2D points.

Parameters

point_a

`[ndarray | RealSequence | Point2D]` *Point2D* class representing the first point.

point_b

`[ndarray | RealSequence | Point2D]` *Point2D* class representing the second point.

Returns

Vector2D

2D vector from `point_a` to `point_b`.

Notes

The resulting 2D vector is always expressed in *Point2D* base units.

UnitVector3D

class `ansys.geometry.core.math.vector.UnitVector3D(shape, dtype=float, buffer=None, offset=0, strides=None, order=None)`

Bases: `Vector3D`

Provides for creating and managing a 3D unit vector.

Parameters

input

`[ndarray | RealSequence | Vector3D]`

- 1D `numpy.ndarray` class with shape(X,)
- `Vector3D`

Overview

Constructors

<code>from_points</code>	Create a 3D unit vector from two distinct 3D points.
--------------------------	--

Import detail

```
from ansys.geometry.core.math.vector import UnitVector3D
```

Method detail

```
classmethod UnitVector3D.from_points(point_a: numpy.ndarray | ansys.geometry.core.typing.RealSequence
                                     | ansys.geometry.core.math.point.Point3D, point_b: numpy.ndarray |
                                     ansys.geometry.core.typing.RealSequence |
                                     ansys.geometry.core.math.point.Point3D)
```

Create a 3D unit vector from two distinct 3D points.

Parameters

point_a

[`ndarray` | `RealSequence` | `Point3D`] `Point3D` class representing the first point.

point_b

[`ndarray` | `RealSequence` | `Point3D`] `Point3D` class representing the second point.

Returns

UnitVector3D

3D unit vector from `point_a` to `point_b`.

UnitVector2D

```
class ansys.geometry.core.math.vector.UnitVector2D(shape, dtype=float, buffer=None, offset=0,
                                                    strides=None, order=None)
```

Bases: `Vector2D`

Provides for creating and managing a 3D unit vector.

Parameters

input

[`ndarray` | `RealSequence` | `Vector2D`]

- 1D `numpy.ndarray` class with shape(X,)
- `Vector2D`

Overview

Constructors

```
from_points Create a 2D unit vector from two distinct 2D points.
```

Import detail

```
from ansys.geometry.core.math.vector import UnitVector2D
```

Method detail

classmethod `UnitVector2D.from_points`(*point_a*: `numpy.ndarray` | `ansys.geometry.core.typing.RealSequence` | `ansys.geometry.core.math.point.Point2D`, *point_b*: `numpy.ndarray` | `ansys.geometry.core.typing.RealSequence` | `ansys.geometry.core.math.point.Point2D`)

Create a 2D unit vector from two distinct 2D points.

Parameters

point_a

[`ndarray` | `RealSequence` | `Point2D`] *Point2D* class representing the first point.

point_b

[`ndarray` | `RealSequence` | `Point2D`] *Point2D* class representing the second point.

Returns

UnitVector2D

2D unit vector from `point_a` to `point_b`.

Description

Provides for creating and managing 2D and 3D vectors.

Description

PyAnsys Geometry math subpackage.

The misc package

Summary

Submodules

<i>accuracy</i>	Provides for evaluating decimal precision.
<i>auxiliary</i>	Auxiliary functions for the PyAnsys Geometry library.
<i>checks</i>	Provides functions for performing common checks.
<i>measurements</i>	Provides various measurement-related classes.
<i>options</i>	Provides various option classes.
<i>units</i>	Provides for handling units homogeneously throughout PyAnsys Geometry.

The accuracy.py module

Summary

Classes

<i>Accuracy</i>	Decimal precision evaluations for math operations.
-----------------	--

Constants

<code>LENGTH_ACCURACY</code>	Constant for decimal accuracy in length comparisons.
<code>ANGLE_ACCURACY</code>	Constant for decimal accuracy in angle comparisons.
<code>DOUBLE_ACCURACY</code>	Constant for double accuracy.

Accuracy

class `ansys.geometry.core.misc.accuracy.Accuracy`

Decimal precision evaluations for math operations.

Overview

Static methods

<code>length_accuracy</code>	Return the <code>LENGTH_ACCURACY</code> constant.
<code>angle_accuracy</code>	Return the <code>ANGLE_ACCURACY</code> constant.
<code>double_accuracy</code>	Return the <code>DOUBLE_ACCURACY</code> constant.
<code>length_is_equal</code>	Check if the comparison length is equal to the reference length.
<code>equal_doubles</code>	Compare two double values.
<code>compare_with_tolerance</code>	Compare two doubles given the relative and absolute tolerances.
<code>length_is_greater_than_or_equal</code>	Check if the length is greater than the reference length.
<code>length_is_less_than_or_equal</code>	Check if the length is less than or equal to the reference length.
<code>length_is_zero</code>	Check if the length is within the length accuracy of exact zero.
<code>length_is_negative</code>	Check if the length is below a negative length accuracy.
<code>length_is_positive</code>	Check if the length is above a positive length accuracy.
<code>angle_is_zero</code>	Check if the length is within the angle accuracy of exact zero.
<code>angle_is_negative</code>	Check if the angle is below a negative angle accuracy.
<code>angle_is_positive</code>	Check if the angle is above a positive angle accuracy.
<code>is_within_tolerance</code>	Check if two values are inside a relative and absolute tolerance.

Import detail

```
from ansys.geometry.core.misc.accuracy import Accuracy
```

Method detail

static `Accuracy.length_accuracy()` → `ansys.geometry.core.typing.Real`

Return the `LENGTH_ACCURACY` constant.

static `Accuracy.angle_accuracy()` → `ansys.geometry.core.typing.Real`

Return the `ANGLE_ACCURACY` constant.

static `Accuracy.double_accuracy()` → `ansys.geometry.core.typing.Real`

Return the `DOUBLE_ACCURACY` constant.

static `Accuracy.length_is_equal(comparison_length: ansys.geometry.core.typing.Real, reference_length: ansys.geometry.core.typing.Real) → bool`

Check if the comparison length is equal to the reference length.

Returns

bool

True if the comparison length is equal to the reference length within the length accuracy, False otherwise.

Notes

The check is done up to the constant value specified for LENGTH_ACCURACY.

static Accuracy.**equal_doubles**(*a: ansys.geometry.core.typing.Real, b: ansys.geometry.core.typing.Real*)
Compare two double values.

static Accuracy.**compare_with_tolerance**(*a: ansys.geometry.core.typing.Real, b: ansys.geometry.core.typing.Real, relative_tolerance: ansys.geometry.core.typing.Real, absolute_tolerance: ansys.geometry.core.typing.Real*) → *ansys.geometry.core.typing.Real*

Compare two doubles given the relative and absolute tolerances.

static Accuracy.**length_is_greater_than_or_equal**(*comparison_length: ansys.geometry.core.typing.Real, reference_length: ansys.geometry.core.typing.Real*) → **bool**

Check if the length is greater than the reference length.

Returns**bool**

True if the comparison length is greater than the reference length within the length accuracy, False otherwise.

Notes

The check is done up to the constant value specified for LENGTH_ACCURACY.

static Accuracy.**length_is_less_than_or_equal**(*comparison_length: ansys.geometry.core.typing.Real, reference_length: ansys.geometry.core.typing.Real*) → **bool**

Check if the length is less than or equal to the reference length.

Returns**bool**

True if the comparison length is less than or equal to the reference length within the length accuracy, False otherwise.

Notes

The check is done up to the constant value specified for LENGTH_ACCURACY.

static Accuracy.**length_is_zero**(*length: ansys.geometry.core.typing.Real*) → **bool**

Check if the length is within the length accuracy of exact zero.

Returns**bool**

True if the length is within the length accuracy of exact zero, False otherwise.

static Accuracy.**length_is_negative**(*length: ansys.geometry.core.typing.Real*) → **bool**

Check if the length is below a negative length accuracy.

Returns

bool**True if the length is below a negative length accuracy,**
False otherwise.**static Accuracy.length_is_positive**(*length: ansys.geometry.core.typing.Real*) → **bool**

Check if the length is above a positive length accuracy.

Returns**bool****True if the length is above a positive length accuracy,**
False otherwise.**static Accuracy.angle_is_zero**(*angle: ansys.geometry.core.typing.Real*) → **bool**

Check if the length is within the angle accuracy of exact zero.

Returns**bool****True if the length is within the angle accuracy of exact zero,**
False otherwise.**static Accuracy.angle_is_negative**(*angle: ansys.geometry.core.typing.Real*) → **bool**

Check if the angle is below a negative angle accuracy.

Returns**bool****True if the angle is below a negative angle accuracy,**
False otherwise.**static Accuracy.angle_is_positive**(*angle: ansys.geometry.core.typing.Real*) → **bool**

Check if the angle is above a positive angle accuracy.

Returns**bool****True if the angle is above a positive angle accuracy,**
False otherwise.**static Accuracy.is_within_tolerance**(*a: ansys.geometry.core.typing.Real, b:*
ansys.geometry.core.typing.Real, relative_tolerance:
ansys.geometry.core.typing.Real, absolute_tolerance:
ansys.geometry.core.typing.Real) → **bool**

Check if two values are inside a relative and absolute tolerance.

Parameters**a**

[Real] First value.

b

[Real] Second value.

relative_tolerance

[Real] Relative tolerance accepted.

absolute_tolerance

[Real] Absolute tolerance accepted.

Returns

bool

True if the values are inside the accepted tolerances, False otherwise.

Description

Provides for evaluating decimal precision.

Module detail

`accuracy.LENGTH_ACCURACY: ansys.geometry.core.typing.Real = 1e-08`

Constant for decimal accuracy in length comparisons.

`accuracy.ANGLE_ACCURACY: ansys.geometry.core.typing.Real = 1e-06`

Constant for decimal accuracy in angle comparisons.

`accuracy.DOUBLE_ACCURACY: ansys.geometry.core.typing.Real = 1e-13`

Constant for double accuracy.

The auxiliary.py module

Summary

Functions

<code>get_design_from_component</code>	Get the Design of the given Component object.
<code>get_design_from_body</code>	Get the Design of the given Body object.
<code>get_design_from_face</code>	Get the Design of the given Face object.
<code>get_design_from_edge</code>	Get the Design of the given Edge object.
<code>get_all_bodies_from_design</code>	Find all the Body objects inside a Design.
<code>get_bodies_from_ids</code>	Find the Body objects inside a Design from its ids.
<code>get_faces_from_ids</code>	Find the Face objects inside a Design from its ids.
<code>get_edges_from_ids</code>	Find the Edge objects inside a Design from its ids.
<code>convert_color_to_hex</code>	Get the hex string color from input formats.

Description

Auxiliary functions for the PyAnsys Geometry library.

Module detail

`auxiliary.get_design_from_component(component: ansys.geometry.core.designer.component.Component)`
 → `ansys.geometry.core.designer.design.Design`

Get the Design of the given Component object.

Parameters

component

[Component] The component object for which to find the Design.

Returns

Design

The Design of the provided component object.

`auxiliary.get_design_from_body`(*body*: ansys.geometry.core.designer.body.Body) →
ansys.geometry.core.designer.design.Design

Get the Design of the given Body object.

Parameters

body
 [Body] The body object for which to find the Design.

Returns

Design
 The Design of the provided body object.

`auxiliary.get_design_from_face`(*face*: ansys.geometry.core.designer.face.Face) →
ansys.geometry.core.designer.design.Design

Get the Design of the given Face object.

Parameters

face
 [Face] The face object for which to find the Design.

Returns

Design
 The Design of the provided face object.

`auxiliary.get_design_from_edge`(*edge*: ansys.geometry.core.designer.edge.Edge) →
ansys.geometry.core.designer.design.Design

Get the Design of the given Edge object.

Parameters

edge
 [Edge] The edge object for which to find the Design.

Returns

Design
 The Design of the provided edge object.

`auxiliary.get_all_bodies_from_design`(*design*: ansys.geometry.core.designer.design.Design) →
list[ansys.geometry.core.designer.body.Body]

Find all the Body objects inside a Design.

Parameters

design
 [Design] Parent design for the bodies.

Returns

list[Body]
 List of Body objects.

Notes

This method takes a design and gets the corresponding Body objects.

`auxiliary.get_bodies_from_ids`(*design*: ansys.geometry.core.designer.design.Design, *body_ids*: list[str]) → list[ansys.geometry.core.designer.body.Body]

Find the Body objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the faces.

body_ids
[list[str]] List of body ids.

Returns

list[Body]
List of Body objects.

Notes

This method takes a design and body ids, and gets their corresponding Body object.

`auxiliary.get_faces_from_ids`(*design*: ansys.geometry.core.designer.design.Design, *face_ids*: list[str]) → list[ansys.geometry.core.designer.face.Face]

Find the Face objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the faces.

face_ids
[list[str]] List of face ids.

Returns

list[Face]
List of Face objects.

Notes

This method takes a design and face ids, and gets their corresponding Face object.

`auxiliary.get_edges_from_ids`(*design*: ansys.geometry.core.designer.design.Design, *edge_ids*: list[str]) → list[ansys.geometry.core.designer.edge.Edge]

Find the Edge objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the edges.

edge_ids
[list[str]] List of edge ids.

Returns

list[Edge]
List of Edge objects.

Notes

This method takes a design and edge ids, and gets their corresponding Edge objects.

auxiliary.**convert_color_to_hex**(color: str | tuple[float, float, float]) → str

Get the hex string color from input formats.

Parameters

color

[str | tuple[float, float, float]] Color to set the body to. This can be a string representing a color name or a tuple of RGB values in the range [0, 1] (RGBA) or [0, 255] (pure RGB).

Returns

str

The hex code string for the color.

The checks.py module

Summary

Functions

<code>ensure_design_is_active</code>	Make sure that the design is active before executing a method.
<code>check_is_float_int</code>	Check if a parameter has a float or integer value.
<code>check_ndarray_is_float_int</code>	Check if a <code>numpy.ndarray</code> has float/integer types.
<code>check_ndarray_is_not_none</code>	Check if a <code>numpy.ndarray</code> is all None.
<code>check_ndarray_is_all_nan</code>	Check if a <code>numpy.ndarray</code> is all nan-valued.
<code>check_ndarray_is_non_zero</code>	Check if a <code>numpy.ndarray</code> is zero-valued.
<code>check_pint_unit_compatibility</code>	Check if input <code>pint.Unit</code> is compatible with the expected input.
<code>check_type_equivalence</code>	Check if an input object is of the same class as an expected object.
<code>check_type</code>	Check if an input object is of the same type as expected types.
<code>check_type_all_elements_in_iterable</code>	Check if all elements in an iterable are of the same type as expected.
<code>min_backend_version</code>	Compare a minimum required version to the current backend version.
<code>deprecated_method</code>	Decorate a method as deprecated.
<code>deprecated_argument</code>	Decorate a method argument as deprecated.

Description

Provides functions for performing common checks.

Module detail

checks.**ensure_design_is_active**(method)

Make sure that the design is active before executing a method.

This function is necessary to be called whenever we do any operation on the design. If we are just accessing information of the class, it is not necessary to call this.

checks.**check_is_float_int**(param: object, param_name: str | None = None) → None

Check if a parameter has a float or integer value.

Parameters

param
[object] Object instance to check.

param_name
[str, default: None] Parameter name (if any).

Raises

TypeError
If the parameter does not have a float or integer value.

`checks.check_ndarray_is_float_int`(*param: numpy.ndarray, param_name: str | None = None*) → None
Check if a `numpy.ndarray` has float/integer types.

Parameters

param
[ndarray] `numpy.ndarray` instance to check.

param_name
[str, default: None] `numpy.ndarray` instance name (if any).

Raises

TypeError
If the `numpy.ndarray` instance does not have float or integer values.

`checks.check_ndarray_is_not_none`(*param: numpy.ndarray, param_name: str | None = None*) → None
Check if a `numpy.ndarray` is all None.

Parameters

param
[ndarray] `numpy.ndarray` instance to check.

param_name
[str, default: None] `numpy.ndarray` instance name (if any).

Raises

ValueError
If the `numpy.ndarray` instance has a value of None for all parameters.

`checks.check_ndarray_is_all_nan`(*param: numpy.ndarray, param_name: str | None = None*) → None
Check if a `numpy.ndarray` is all nan-valued.

Parameters

param
[ndarray] `numpy.ndarray` instance to check.

param_name
[str or None, default: None] `numpy.ndarray` instance name (if any).

Raises

ValueError
If the `numpy.ndarray` instance is all nan-valued.

`checks.check_ndarray_is_non_zero`(*param: numpy.ndarray, param_name: str | None = None*) → None
Check if a `numpy.ndarray` is zero-valued.

Parameters

param
[ndarray] `numpy.ndarray` instance to check.

param_name
[str, default: None] `numpy.ndarray` instance name (if any).

Raises

ValueError
If the `numpy.ndarray` instance is zero-valued.

`checks.check_pint_unit_compatibility(input: pint.Unit, expected: pint.Unit) → None`

Check if input `pint.Unit` is compatible with the expected input.

Parameters

input
[Unit] `pint.Unit` input.

expected
[Unit] `pint.Unit` expected dimensionality.

Raises

TypeError
If the input is not compatible with the `pint.Unit` class.

`checks.check_type_equivalence(input: object, expected: object) → None`

Check if an input object is of the same class as an expected object.

Parameters

input
[object] Input object.

expected
[object] Expected object.

Raises

TypeError
If the objects are not of the same class.

`checks.check_type(input: object, expected_type: type | tuple[type, Ellipsis]) → None`

Check if an input object is of the same type as expected types.

Parameters

input
[object] Input object.

expected_type
[*type* | *tuple*[*type*, ...]] One or more types to compare the input object against.

Raises

TypeError
If the object does not match the one or more expected types.

`checks.check_type_all_elements_in_iterable(input: collections.abc.Iterable, expected_type: type | tuple[type, Ellipsis]) → None`

Check if all elements in an iterable are of the same type as expected.

Parameters

input

[Iterable] Input iterable.

expected_type

[type | tuple[type, ...]] One or more types to compare the input object against.

Raises**TypeError**

If one of the elements in the iterable does not match the one or more expected types.

`checks.min_backend_version(major: int, minor: int, service_pack: int)`

Compare a minimum required version to the current backend version.

Parameters**major**

[int] Minimum major version required by the method.

minor

[int] Minimum minor version required by the method.

service_pack

[int] Minimum service pack version required by the method.

Raises**GeometryRuntimeError**

If the method version is higher than the backend version.

GeometryRuntimeError

If the client is not available.

`checks.deprecated_method(alternative: str | None = None, info: str | None = None, version: str | None = None, remove: str | None = None)`

Decorate a method as deprecated.

Parameters**alternative**

[str, default: None] Alternative method to use. If provided, the warning message will include the alternative method.

info

[str, default: None] Additional information to include in the warning message.

version

[str, default: None] Version where the method was deprecated.

remove

[str, default: None] Version where the method will be removed.

`checks.deprecated_argument(arg: str, alternative: str | None = None, info: str | None = None, version: str | None = None, remove: str | None = None)`

Decorate a method argument as deprecated.

Parameters**arg**

[str] Argument to mark as deprecated.

alternative

[str, default: None] Alternative argument to use. If provided, the warning message will include the alternative argument.

info

[str, default: None] Additional information to include in the warning message.

version

[str, default: None] Version where the method was deprecated.

remove

[str, default: None] Version where the method will be removed.

The measurements.py module

Summary

Classes

<i>SingletonMeta</i>	Provides a thread-safe implementation of a singleton design pattern.
<i>DefaultUnitsClass</i>	Provides default units for PyAnsys Geometry.
<i>Measurement</i>	Provides the <code>PhysicalQuantity</code> subclass for holding a measurement.
<i>Distance</i>	Provides the <code>Measurement</code> subclass for holding a distance.
<i>Angle</i>	Provides the <code>Measurement</code> subclass for holding an angle.

Constants

<code>DEFAULT_UNITS</code>	PyAnsys Geometry default units object.
----------------------------	--

SingletonMeta

class `ansys.geometry.core.misc.measurements.SingletonMeta`

Bases: `type`

Provides a thread-safe implementation of a singleton design pattern.

Overview

Special methods

<code>__call__</code>	Return a single instance of the class.
-----------------------	--

Import detail

```
from ansys.geometry.core.misc.measurements import SingletonMeta
```

Method detail

`SingletonMeta.__call__(*args, **kwargs)`

Return a single instance of the class.

Possible changes to the value of the `__init__` argument do not affect the returned instance.

DefaultUnitsClass

class ansys.geometry.core.misc.measurements.DefaultUnitsClass

Provides default units for PyAnsys Geometry.

Overview

Properties

<i>LENGTH</i>	Default length unit for PyAnsys Geometry.
<i>ANGLE</i>	Default angle unit for PyAnsys Geometry.
<i>SERVER_LENGTH</i>	Default length unit for gRPC messages.
<i>SERVER_AREA</i>	Default area unit for gRPC messages.
<i>SERVER_VOLUME</i>	Default volume unit for gRPC messages.
<i>SERVER_ANGLE</i>	Default angle unit for gRPC messages.

Import detail

```
from ansys.geometry.core.misc.measurements import DefaultUnitsClass
```

Property detail

property DefaultUnitsClass.**LENGTH**: **pint.Unit**

Default length unit for PyAnsys Geometry.

property DefaultUnitsClass.**ANGLE**: **pint.Unit**

Default angle unit for PyAnsys Geometry.

property DefaultUnitsClass.**SERVER_LENGTH**: **pint.Unit**

Default length unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

property DefaultUnitsClass.**SERVER_AREA**: **pint.Unit**

Default area unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

property DefaultUnitsClass.**SERVER_VOLUME**: **pint.Unit**

Default volume unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

property DefaultUnitsClass.**SERVER_ANGLE**: **pint.Unit**

Default angle unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

Measurement

```
class ansys.geometry.core.misc.measurements.Measurement(value: ansys.geometry.core.typing.Real | pint.Quantity, unit: pint.Unit, dimensions: pint.Unit)
```

Bases: `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides the `PhysicalQuantity` subclass for holding a measurement.

Parameters

value

[`Real` | `Quantity`] Value of the measurement.

unit

[`Unit`] Units for the measurement.

dimensions

[`Unit`] Units for extracting the dimensions of the measurement. If `~pint.Unit.meter` is given, the dimension extracted is [`length`].

Overview

Properties

<code>value</code>	Value of the measurement.
--------------------	---------------------------

Special methods

<code>__repr__</code>	Representation of the Measurement class.
<code>__eq__</code>	Equals operator for the Measurement class.

Import detail

```
from ansys.geometry.core.misc.measurements import Measurement
```

Property detail

property `Measurement.value: pint.Quantity`

Value of the measurement.

Method detail

`Measurement.__repr__()`

Representation of the Measurement class.

`Measurement.__eq__(other: Measurement) → bool`

Equals operator for the Measurement class.

Distance

```
class ansys.geometry.core.misc.measurements.Distance(value: ansys.geometry.core.typing.Real |  
                                                    pint.Quantity, unit: pint.Unit | None = None)
```

Bases: Measurement

Provides the Measurement subclass for holding a distance.

Parameters

value

[Real | Quantity] Value of the distance.

unit

[Unit, default: DEFAULT_UNITS.LENGTH] Units for the distance.

Import detail

```
from ansys.geometry.core.misc.measurements import Distance
```

Angle

```
class ansys.geometry.core.misc.measurements.Angle(value: ansys.geometry.core.typing.Real |  
                                                    pint.Quantity, unit: pint.Unit | None = None)
```

Bases: Measurement

Provides the Measurement subclass for holding an angle.

Parameters

value

[Real | Quantity] Value of the angle.

unit

[Unit, default: DEFAULT_UNITS.ANGLE] Units for the distance.

Import detail

```
from ansys.geometry.core.misc.measurements import Angle
```

Description

Provides various measurement-related classes.

Module detail

measurements.**DEFAULT_UNITS**

PyAnsys Geometry default units object.

The options.py module

Summary

Classes

<i>ImportOptions</i> Import options when opening a file.
--

ImportOptions

class `ansys.geometry.core.misc.options.ImportOptions`

Import options when opening a file.

Parameters

cleanup_bodies

[`bool = False`] Simplify geometry and clean up topology.

import_coordinate_systems

[`bool = False`] Import coordinate systems.

import_curves

[`bool = False`] Import curves.

import_hidden_components_and_geometry

[`bool = False`] Import hidden components and geometry.

import_names

[`bool = False`] Import names of bodies and curves.

import_planes

[`bool = False`] Import planes.

import_points

[`bool = False`] Import points.

Overview

Methods

<code>to_dict</code>	Provide the dictionary representation of the ImportOptions class.
----------------------	---

Attributes

<code>cleanup_bodies</code>
<code>import_coordinate_systems</code>
<code>import_curves</code>
<code>import_hidden_components_and_geometry</code>
<code>import_names</code>
<code>import_planes</code>
<code>import_points</code>

Import detail

```
from ansys.geometry.core.misc.options import ImportOptions
```

Attribute detail

`ImportOptions.cleanup_bodies`: `bool = False`

`ImportOptions.import_coordinate_systems`: `bool = False`

`ImportOptions.import_curves: bool = False`

`ImportOptions.import_hidden_components_and_geometry: bool = False`

`ImportOptions.import_names: bool = False`

`ImportOptions.import_planes: bool = False`

`ImportOptions.import_points: bool = False`

Method detail

`ImportOptions.to_dict()`

Provide the dictionary representation of the `ImportOptions` class.

Description

Provides various option classes.

The `units.py` module

Summary

Classes

<i>PhysicalQuantity</i>	Provides the base class for handling units throughout PyAnsys Geometry.
-------------------------	---

Constants

<i>UNITS</i>	Units manager.
--------------	----------------

PhysicalQuantity

class `ansys.geometry.core.misc.units.PhysicalQuantity`(*unit: pint.Unit, expected_dimensions: pint.Unit | None = None*)

Provides the base class for handling units throughout PyAnsys Geometry.

Parameters

unit

[`Unit`] Units for the class.

expected_dimensions

[`Unit`, default: `None`] Units for the dimensionality of the physical quantity.

Overview

Properties

<i>unit</i>	Unit of the object.
<i>base_unit</i>	Base unit of the object.

Import detail

```
from ansys.geometry.core.misc.units import PhysicalQuantity
```

Property detail

property `PhysicalQuantity.unit: pint.Unit`

Unit of the object.

property `PhysicalQuantity.base_unit: pint.Unit`

Base unit of the object.

Description

Provides for handling units homogeneously throughout PyAnsys Geometry.

Module detail

`units.UNITS`

Units manager.

Description

Provides the PyAnsys Geometry miscellaneous subpackage.

The parameters package

Summary

Submodules

<code>parameter</code>	Provides get and set methods for parameters.
------------------------	--

The parameter.py module

Summary

Classes

<code>Parameter</code>	Represents a parameter.
------------------------	-------------------------

Enums

<code>ParameterType</code>	Provides values for the parameter types supported.
<code>ParameterUpdateStatus</code>	Provides values for the status messages associated with parameter updates.

Parameter


```
class ansys.geometry.core.parameters.parameter.Parameter(id: int, name: str, dimension_type:
ParameterType, dimension_value:
ansys.geometry.core.typing.Real)
```

Represents a parameter.

Parameters

- id**
[int] Unique ID for the parameter.
- name**
[str] Name of the parameter.
- dimension_type**
[ParameterType] Type of the parameter.
- dimension_value**
[float] Value of the parameter.

Overview

Properties

<i>name</i>	Get the name of the parameter.
<i>dimension_value</i>	Get the value of the parameter.
<i>dimension_type</i>	Get the type of the parameter.

Attributes

<i>id</i>

Import detail

```
from ansys.geometry.core.parameters.parameter import Parameter
```

Property detail

- property** Parameter.**name**: str
Get the name of the parameter.
- property** Parameter.**dimension_value**: ansys.geometry.core.typing.Real
Get the value of the parameter.
- property** Parameter.**dimension_type**: ParameterType
Get the type of the parameter.

Attribute detail

Parameter.**id**

ParameterType

```
class ansys.geometry.core.parameters.parameter.ParameterType(*args, **kwargs)
```

Bases: `enum.Enum`

Provides values for the parameter types supported.

Overview

Attributes

<code>DIMENSIONTYPE_UNKNOWN</code>
<code>DIMENSIONTYPE_LINEAR</code>
<code>DIMENSIONTYPE_DIAMETRIC</code>
<code>DIMENSIONTYPE_RADIAL</code>
<code>DIMENSIONTYPE_ARC</code>
<code>DIMENSIONTYPE_AREA</code>
<code>DIMENSIONTYPE_VOLUME</code>
<code>DIMENSIONTYPE_MASS</code>
<code>DIMENSIONTYPE_ANGULAR</code>
<code>DIMENSIONTYPE_COUNT</code>
<code>DIMENSIONTYPE_UNITLESS</code>

Import detail

```
from ansys.geometry.core.parameters.parameter import ParameterType
```

Attribute detail

```
ParameterType.DIMENSIONTYPE_UNKNOWN = 0
```

```
ParameterType.DIMENSIONTYPE_LINEAR = 1
```

```
ParameterType.DIMENSIONTYPE_DIAMETRIC = 2
```

```
ParameterType.DIMENSIONTYPE_RADIAL = 3
```

```
ParameterType.DIMENSIONTYPE_ARC = 4
```

```
ParameterType.DIMENSIONTYPE_AREA = 5
```

```
ParameterType.DIMENSIONTYPE_VOLUME = 6
```

```
ParameterType.DIMENSIONTYPE_MASS = 7
```

```
ParameterType.DIMENSIONTYPE_ANGULAR = 8
```

```
ParameterType.DIMENSIONTYPE_COUNT = 9
```

```
ParameterType.DIMENSIONTYPE_UNITLESS = 10
```

ParameterUpdateStatus

class ansys.geometry.core.parameters.parameter.ParameterUpdateStatus(*args, **kwargs)

Bases: enum.Enum

Provides values for the status messages associated with parameter updates.

Overview

Attributes

<i>SUCCESS</i>
<i>FAILURE</i>
<i>CONSTRAINED_PARAMETERS</i>
<i>UNKNOWN</i>

Import detail

```
from ansys.geometry.core.parameters.parameter import ParameterUpdateStatus
```

Attribute detail

ParameterUpdateStatus.SUCCESS = 0

ParameterUpdateStatus.FAILURE = 1

ParameterUpdateStatus.CONSTRAINED_PARAMETERS = 2

ParameterUpdateStatus.UNKNOWN = 3

Description

Provides get and set methods for parameters.

Description

PyAnsys Geometry parameters subpackage.

The plotting package

Summary

Subpackages

<i>widgets</i>	Submodule providing widgets for the PyAnsys Geometry plotter.
----------------	---

Submodules

<i>plotter</i>	Provides plotting for various PyAnsys Geometry objects.
----------------	---

The widgets package

Summary

Submodules

<code>show_design_point</code>	Provides the ruler widget for the PyAnsys Geometry plotter.
--------------------------------	---

The `show_design_point.py` module

Summary

Classes

<code>ShowDesignPoints</code>	Provides the a button to hide/show DesignPoint objects in the plotter.
-------------------------------	--

ShowDesignPoints

```
class ansys.geometry.core.plotting.widgets.show_design_point.ShowDesignPoints(plotter_helper:
                                                                              an-
                                                                              sys.tools.visualization_interface.l)
```

Bases: `ansys.tools.visualization_interface.backends.pyvista.widgets.PlotterWidget`

Provides the a button to hide/show DesignPoint objects in the plotter.

Parameters

`plotter_helper`

[`GeometryPlotter`] Provides the plotter to add the button to.

Overview

Methods

<code>callback</code>	Remove or add the DesignPoint actors upon click.
<code>update</code>	Define the configuration and representation of the button widget.

Attributes

<code>plotter_helper</code>

Import detail

```
from ansys.geometry.core.plotting.widgets.show_design_point import ShowDesignPoints
```

Attribute detail

`ShowDesignPoints.plotter_helper`

Method detail

ShowDesignPoints.**callback**(state: *bool*) → *None*

Remove or add the DesignPoint actors upon click.

Parameters

state

[*bool*] State of the button, which is inherited from PyVista. The value is *True* if the button is active.

ShowDesignPoints.**update**() → *None*

Define the configuration and representation of the button widget.

Description

Provides the ruler widget for the PyAnsys Geometry plotter.

Description

Submodule providing widgets for the PyAnsys Geometry plotter.

The `plotter.py` module

Summary

Classes

<i>GeometryPlotter</i> Plotter for PyAnsys Geometry objects.
--

Constants

<i>POLYDATA_COLOR_CYCLER</i>

GeometryPlotter

```
class ansys.geometry.core.plotting.plotter.GeometryPlotter(use_trame: bool | None = None,  
                                                         use_service_colors: bool | None =  
                                                         None, allow_picking: bool = False,  
                                                         show_plane: bool = True)
```

Bases: `ansys.tools.visualization_interface.Plotter`

Plotter for PyAnsys Geometry objects.

This class is an implementation of the `PlotterInterface` class.

Parameters

use_trame

[*bool*, optional] Whether to use trame visualizer or not, by default *None*.

use_service_colors

[*bool*, optional] Whether to use service colors or not, by default *None*.

allow_picking

[*bool*, optional] Whether to allow picking or not, by default *False*.

show_plane

[bool, optional] Whether to show the plane in the scene, by default True.

Overview**Methods**

<code>add_frame</code>	Plot a frame in the scene.
<code>add_plane</code>	Plot a plane in the scene.
<code>add_sketch</code>	Plot a sketch in the scene.
<code>add_body_edges</code>	Add the outer edges of a body to the plot.
<code>add_body</code>	Add a body to the scene.
<code>add_face</code>	Add a face to the scene.
<code>add_component</code>	Add a component to the scene.
<code>add_component_by_body</code>	Add a component on a per body basis.
<code>add_sketch_polydata</code>	Add sketches to the scene from PyVista polydata.
<code>add_design_point</code>	Add a DesignPoint object to the plotter.
<code>plot_iter</code>	Add a list of any type of object to the scene.
<code>plot</code>	Add a custom mesh to the plotter.
<code>show</code>	Show the plotter.
<code>export_glb</code>	Export the design to a glb file. Does not support picked objects.

Properties

<code>use_service_colors</code>	Indicates whether to use service colors for plotting purposes.
---------------------------------	--

Import detail

```
from ansys.geometry.core.plotting.plotter import GeometryPlotter
```

Property detail

property `GeometryPlotter.use_service_colors`: **bool**

Indicates whether to use service colors for plotting purposes.

Method detail

`GeometryPlotter.add_frame`(*frame*: `ansys.geometry.core.math.frame.Frame`, *plotting_options*: `dict` | `None` = `None`) → `None`

Plot a frame in the scene.

Parameters**frame**

[Frame] Frame to render in the scene.

plotting_options

[dict, default: `None`] dictionary containing parameters accepted by the `pyvista.create_axes_marker()` class for customizing the frame rendering in the scene.

`GeometryPlotter.add_plane`(*plane*: `ansys.geometry.core.math.plane.Plane`, *plane_options*: `dict` | `None` = `None`, *plotting_options*: `dict` | `None` = `None`) → `None`

Plot a plane in the scene.

Parameters

plane

[Plane] Plane to render in the scene.

plane_options

[dict, default: None] dictionary containing parameters accepted by the `pyvista.Plane` function for customizing the mesh representing the plane.

plotting_options

[dict, default: None] dictionary containing parameters accepted by the `Plotter.add_mesh` method for customizing the mesh rendering of the plane.

`GeometryPlotter.add_sketch`(*sketch*: `ansys.geometry.core.sketch.sketch.Sketch`, *show_plane*: `bool = False`, *show_frame*: `bool = False`, ***plotting_options*: `dict | None`) → None

Plot a sketch in the scene.

Parameters

sketch

[Sketch] Sketch to render in the scene.

show_plane

[bool, default: False] Whether to render the sketch plane in the scene.

show_frame

[bool, default: False] Whether to show the frame in the scene.

**plotting_options

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_body_edges`(*body_plot*: `ansys.tools.visualization_interface.MeshObjectPlot`, ***plotting_options*: `dict | None`) → None

Add the outer edges of a body to the plot.

This method has the side effect of adding the edges to the `GeomObject` that you pass through the parameters.

Parameters

body_plot

[MeshObjectPlot] Body of which to add the edges.

**plotting_options

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_body`(*body*: `ansys.geometry.core.designer.body.Body`, *merge*: `bool = False`, ***plotting_options*: `dict | None`) → None

Add a body to the scene.

Parameters

body

[Body] Body to add.

merge

[bool, default: False] Whether to merge the body into a single mesh. When True, the individual faces of the tessellation are merged. This preserves the number of triangles and only merges the topology.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_face`(*face*: ansys.geometry.core.designer.face.Face, ***plotting_options*: dict | None) → None

Add a face to the scene.

Parameters**face**

[Face] Face to add.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_component`(*component*: ansys.geometry.core.designer.component.Component, *merge_component*: bool = False, *merge_bodies*: bool = False, ***plotting_options*) → None

Add a component to the scene.

Parameters**component**

[Component] Component to add.

merge_component

[bool, default: False] Whether to merge the component into a single dataset. When True, all the individual bodies are effectively combined into a single dataset without any hierarchy.

merge_bodies

[bool, default: False] Whether to merge each body into a single dataset. When True, all the faces of each individual body are effectively combined into a single dataset without separating faces.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_component_by_body`(*component*: ansys.geometry.core.designer.component.Component, *merge_bodies*: bool, ***plotting_options*: dict | None) → None

Add a component on a per body basis.

Parameters**component**

[Component] Component to add.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

Notes

This will allow to make use of the service colors. At the same time, it will be slower than the `add_component` method.

`GeometryPlotter.add_sketch_polydata`(*polydata_entries*: list[pyvista.PolyData], *sketch*: ansys.geometry.core.sketch.sketch.Sketch = None, ***plotting_options*) → None

Add sketches to the scene from PyVista polydata.

Parameters

polydata_entries

[list[pyvista.PolyData]] Polydata to add.

sketch

[Sketch, default: None] Sketch to add.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_design_point`(*design_point*: ansys.geometry.core.designer.designpoint.DesignPoint, ***plotting_options*) → None

Add a DesignPoint object to the plotter.

Parameters

design_point

[DesignPoint] DesignPoint to add.

`GeometryPlotter.plot_iter`(*plotting_list*: list[Any], *name_filter*: str = None, ***plotting_options*) → None

Add a list of any type of object to the scene.

These types of objects are supported: Body, Component, list[pv.PolyData], pv.MultiBlock, and Sketch.

Parameters

plotting_list

[list[Any]] list of objects you want to plot.

name_filter

[str, default: None] Regular expression with the desired name or names you want to include in the plotter.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.plot`(*plottable_object*: Any, *name_filter*: str = None, ***plotting_options*) → None

Add a custom mesh to the plotter.

Parameters

plottable_object

[str, default: None] Regular expression with the desired name or names you want to include in the plotter.

name_filter: str, default: None

Regular expression with the desired name or names you want to include in the plotter.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, depend of the backend implementation you are using.

`GeometryPlotter.show`(*plotting_object*: Any = None, *screenshot*: str | None = None, ***plotting_options*) → None | list[Any]

Show the plotter.

Parameters

plotting_object

[Any, default: `None`] Object you can add to the plotter.

screenshot

[`str`, default: `None`] Path to save a screenshot of the plotter.

****plotting_options**

[`dict`, default: `None`] Keyword arguments for the plotter. Arguments depend of the backend implementation you are using.

`GeometryPlotter.export_glb(plotting_object: Any = None, filename: str | pathlib.Path | None = None, **plotting_options) → pathlib.Path`

Export the design to a glb file. Does not support picked objects.

Parameters**plotting_object**

[Any, default: `None`] Object you can add to the plotter.

filename

[`str` | `Path`, default: `None`] Path to save a GLB file of the plotter. If `None`, the file will be saved as `temp_glb.glb`.

****plotting_options**

[`dict`, default: `None`] Keyword arguments for the plotter. Arguments depend of the backend implementation you are using.

Returns**Path**

Path to the exported glb file.

Description

Provides plotting for various PyAnsys Geometry objects.

Module detail

`plotter.POLYDATA_COLOR_CYCLER`

Description

Provides the PyAnsys Geometry plotting subpackage.

The shapes package**Summary****Subpackages**

<code>curves</code>	Provides the PyAnsys Geometry <code>curves</code> subpackage.
<code>surfaces</code>	Provides the PyAnsys Geometry <code>surface</code> subpackage.

Submodules

<i>box_uv</i>	Provides the BoxUV class.
<i>parameterization</i>	Provides the parametrization-related classes.

The curves package

Summary

Submodules

<i>circle</i>	Provides for creating and managing a circle.
<i>curve</i>	Provides the Curve class.
<i>curve_evaluation</i>	Provides for creating and managing a curve.
<i>ellipse</i>	Provides for creating and managing an ellipse.
<i>line</i>	Provides for creating and managing a line.
<i>nurbs</i>	Provides for creating and managing a NURBS curve.
<i>trimmed_curve</i>	Trimmed curve class.

The circle.py module

Summary

Classes

<i>Circle</i>	Provides 3D circle representation.
<i>CircleEvaluation</i>	Provides evaluation of a circle at a given parameter.

Circle

```
class ansys.geometry.core.shapes.curves.circle.Circle(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D,
    radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real, reference:
    numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_X, axis: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes.curves.curve.Curve*

Provides 3D circle representation.

Parameters

origin

[[ndarray](#) | [RealSequence](#) | [Point3D](#)] Origin of the circle.

radius

[[Quantity](#) | [Distance](#) | [Real](#)] Radius of the circle.

reference

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] X-axis direction.

axis

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] Z-axis direction.

Overview**Abstract methods**

<i>contains_param</i>	Check a parameter is within the parametric range of the curve.
<i>contains_point</i>	Check a point is contained by the curve.

Methods

<i>evaluate</i>	Evaluate the circle at a given parameter.
<i>transformed_copy</i>	Create a transformed copy of the circle from a transformation matrix.
<i>mirrored_copy</i>	Create a mirrored copy of the circle along the y-axis.
<i>project_point</i>	Project a point onto the circle and evaluate the circle.
<i>is_coincident_circle</i>	Determine if the circle is coincident with another.
<i>parameterization</i>	Get the parametrization of the circle.

Properties

<i>origin</i>	Origin of the circle.
<i>radius</i>	Radius of the circle.
<i>diameter</i>	Diameter of the circle.
<i>perimeter</i>	Perimeter of the circle.
<i>area</i>	Area of the circle.
<i>dir_x</i>	X-direction of the circle.
<i>dir_y</i>	Y-direction of the circle.
<i>dir_z</i>	Z-direction of the circle.

Special methods

<code>__eq__</code>	Equals operator for the Circle class.
---------------------	---------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.circle import Circle
```

Property detail

property `Circle.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the circle.

property `Circle.radius`: `pint.Quantity`

Radius of the circle.

property `Circle.diameter`: `pint.Quantity`

Diameter of the circle.

property `Circle.perimeter`: `pint.Quantity`

Perimeter of the circle.

property `Circle.area`: `pint.Quantity`

Area of the circle.

property `Circle.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the circle.

property `Circle.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the circle.

property `Circle.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the circle.

Method detail

`Circle.__eq__(other: Circle) → bool`

Equals operator for the Circle class.

`Circle.evaluate(parameter: ansys.geometry.core.typing.Real) → CircleEvaluation`

Evaluate the circle at a given parameter.

Parameters

parameter

[Real] Parameter to evaluate the circle at.

Returns

CircleEvaluation

Resulting evaluation.

`Circle.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Circle`

Create a transformed copy of the circle from a transformation matrix.

Parameters

matrix

[Matrix44] 4x4 transformation matrix to apply to the circle.

Returns

Circle

New circle that is the transformed copy of the original circle.

`Circle.mirrored_copy() → Circle`

Create a mirrored copy of the circle along the y-axis.

Returns

Circle

A new circle that is a mirrored copy of the original circle.

`Circle.project_point(point: ansys.geometry.core.math.point.Point3D) → CircleEvaluation`

Project a point onto the circle and evaluate the circle.

Parameters**point**

[Point3D] Point to project onto the circle.

Returns**CircleEvaluation**

Resulting evaluation.

`Circle.is_coincident_circle(other: Circle) → bool`

Determine if the circle is coincident with another.

Parameters**other**

[Circle] Circle to determine coincidence with.

Returns**bool**

True if this circle is coincident with the other, False otherwise.

`Circle.parameterization() → ansys.geometry.core.shapes.parameterization.Parameterization`

Get the parametrization of the circle.

The parameter of a circle specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

Returns**Parameterization**

Information about how the circle is parameterized.

abstract `Circle.contains_param(param: ansys.geometry.core.typing.Real) → bool`

Check a parameter is within the parametric range of the curve.

abstract `Circle.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

CircleEvaluation

class `ansys.geometry.core.shapes.curves.circle.CircleEvaluation(circle: Circle, parameter: ansys.geometry.core.typing.Real)`

Bases: `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Provides evaluation of a circle at a given parameter.

Parameters

circle: `~ansys.geometry.core.shapes.curves.circle.Circle`

Circle to evaluate.

parameter: `Real`

Parameter to evaluate the circle at.

Overview

Properties

<i>circle</i>	Circle being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation.
<i>normal</i>	Normal to the circle.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the circle.

Import detail

```
from ansys.geometry.core.shapes.curves.circle import CircleEvaluation
```

Property detail

property `CircleEvaluation.circle`: *Circle*

Circle being evaluated.

property `CircleEvaluation.parameter`: `ansys.geometry.core.typing.Real`

Parameter that the evaluation is based upon.

property `CircleEvaluation.position`: `ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

Point3D

Point that lies on the circle at this evaluation.

property `CircleEvaluation.tangent`: `ansys.geometry.core.math.vector.UnitVector3D`

Tangent of the evaluation.

Returns

UnitVector3D

Tangent unit vector to the circle at this evaluation.

property `CircleEvaluation.normal`: `ansys.geometry.core.math.vector.UnitVector3D`

Normal to the circle.

Returns

UnitVector3D

Normal unit vector to the circle at this evaluation.

property `CircleEvaluation.first_derivative`: `ansys.geometry.core.math.vector.Vector3D`

First derivative of the evaluation.

The first derivative is in the direction of the tangent and has a magnitude equal to the velocity (rate of change of position) at that point.

Returns

Vector3D

First derivative of the evaluation.

property CircleEvaluation.**second_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Second derivative of the evaluation.

Returns

Vector3D

Second derivative of the evaluation.

property CircleEvaluation.**curvature**: *ansys.geometry.core.typing.Real*

Curvature of the circle.

Returns

Real

Curvature of the circle.

Description

Provides for creating and managing a circle.

The curve.py module

Summary

Classes

<i>Curve</i>	Provides the abstract base class representing a 3D curve.
--------------	---

Curve

class *ansys.geometry.core.shapes.curves.curve.Curve*

Bases: *abc.ABC*

Provides the abstract base class representing a 3D curve.

Overview

Abstract methods

<i>parameterization</i>	Parameterize the curve.
<i>contains_param</i>	Check a parameter is within the parametric range of the curve.
<i>contains_point</i>	Check a point is contained by the curve.
<i>transformed_copy</i>	Create a transformed copy of the curve.
<i>__eq__</i>	Determine if two curves are equal.
<i>evaluate</i>	Evaluate the curve at the given parameter.
<i>project_point</i>	Project a point to the curve.

Methods

<i>trim</i>	Trim this curve by bounding it with an interval.
-------------	--

Import detail

```
from ansys.geometry.core.shapes.curves.curve import Curve
```

Method detail

abstract `Curve.parameterization()` → *ansys.geometry.core.shapes.parameterization.Parameterization*
Parameterize the curve.

abstract `Curve.contains_param(param: ansys.geometry.core.typing.Real)` → `bool`
Check a parameter is within the parametric range of the curve.

abstract `Curve.contains_point(point: ansys.geometry.core.math.point.Point3D)` → `bool`
Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

abstract `Curve.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44)` → *Curve*
Create a transformed copy of the curve.

abstract `Curve.__eq__(other: Curve)` → `bool`
Determine if two curves are equal.

abstract `Curve.evaluate(parameter: ansys.geometry.core.typing.Real)` →
ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation
Evaluate the curve at the given parameter.

abstract `Curve.project_point(point: ansys.geometry.core.math.point.Point3D)` →
ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation
Project a point to the curve.
This method returns the evaluation at the closest point.

`Curve.trim(interval: ansys.geometry.core.shapes.parameterization.Interval)` →
ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve
Trim this curve by bounding it with an interval.

Returns

TrimmedCurve

The resulting bounded curve.

Description

Provides the Curve class.

The curve_evaluation.py module

Summary

Classes

<i>CurveEvaluation</i>	Provides for evaluating a curve.
------------------------	----------------------------------

CurveEvaluation

class `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`(*parameter: ansys.geometry.core.typing.Real = None*)

Provides for evaluating a curve.

Overview

Methods

<code>is_set</code>	Determine if the parameter for the evaluation has been set.
---------------------	---

Properties

<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>first_derivative</code>	First derivative of the evaluation.
<code>second_derivative</code>	Second derivative of the evaluation.
<code>curvature</code>	Curvature of the evaluation.

Import detail

```
from ansys.geometry.core.shapes.curves.curve_evaluation import CurveEvaluation
```

Property detail

property `CurveEvaluation.parameter: ansys.geometry.core.typing.Real`

Abstractmethod

Parameter that the evaluation is based upon.

property `CurveEvaluation.position: ansys.geometry.core.math.point.Point3D`

Abstractmethod

Position of the evaluation.

property `CurveEvaluation.first_derivative: ansys.geometry.core.math.vector.Vector3D`

Abstractmethod

First derivative of the evaluation.

property `CurveEvaluation.second_derivative: ansys.geometry.core.math.vector.Vector3D`

Abstractmethod

Second derivative of the evaluation.

property `CurveEvaluation.curvature: ansys.geometry.core.typing.Real`

Abstractmethod

Curvature of the evaluation.

Method detail

`CurveEvaluation.is_set()` → `bool`

Determine if the parameter for the evaluation has been set.

Description

Provides for creating and managing a curve.

The `ellipse.py` module

Summary

Classes

<i>Ellipse</i>	Provides 3D ellipse representation.
<i>EllipseEvaluation</i>	Evaluate an ellipse at a given parameter.

Ellipse

```

class ansys.geometry.core.shapes.curves.ellipse.Ellipse(origin: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.point.Point3D,
major_radius: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real,
minor_radius: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real,
reference: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
an-
sys.geometry.core.math.vector.UnitVector3D
|
ansys.geometry.core.math.vector.Vector3D
= UNITVECTOR3D_X, axis:
numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
an-
sys.geometry.core.math.vector.UnitVector3D
|
ansys.geometry.core.math.vector.Vector3D
= UNITVECTOR3D_Z)

```

Bases: `ansys.geometry.core.shapes.curves.curve.Curve`

Provides 3D ellipse representation.

Parameters

origin

[`ndarray` | `RealSequence` | `Point3D`] Origin of the ellipse.

major_radius

[`Quantity` | `Distance` | `Real`] Major radius of the ellipse.

minor_radius

[Quantity | Distance | Real] Minor radius of the ellipse.

reference

[ndarray | RealSequence | UnitVector3D | Vector3D] X-axis direction.

axis

[ndarray | RealSequence | UnitVector3D | Vector3D] Z-axis direction.

Overview**Abstract methods**

<i>contains_param</i>	Check a parameter is within the parametric range of the curve.
<i>contains_point</i>	Check a point is contained by the curve.

Methods

<i>mirrored_copy</i>	Create a mirrored copy of the ellipse along the y-axis.
<i>evaluate</i>	Evaluate the ellipse at the given parameter.
<i>project_point</i>	Project a point onto the ellipse and evaluate the ellipse.
<i>is_coincident_ellipse</i>	Determine if this ellipse is coincident with another.
<i>transformed_copy</i>	Create a transformed copy of the ellipse from a transformation matrix.
<i>parameterization</i>	Get the parametrization of the ellipse.

Properties

<i>origin</i>	Origin of the ellipse.
<i>major_radius</i>	Major radius of the ellipse.
<i>minor_radius</i>	Minor radius of the ellipse.
<i>dir_x</i>	X-direction of the ellipse.
<i>dir_y</i>	Y-direction of the ellipse.
<i>dir_z</i>	Z-direction of the ellipse.
<i>eccentricity</i>	Eccentricity of the ellipse.
<i>linear_eccentricity</i>	Linear eccentricity of the ellipse.
<i>semi_latus_rectum</i>	Semi-latus rectum of the ellipse.
<i>perimeter</i>	Perimeter of the ellipse.
<i>area</i>	Area of the ellipse.

Special methods

<code>__eq__</code>	Equals operator for the Ellipse class.
---------------------	--

Import detail

```
from ansys.geometry.core.shapes.curves.ellipse import Ellipse
```

Property detail

property `Ellipse.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the ellipse.

property `Ellipse.major_radius`: `pint.Quantity`

Major radius of the ellipse.

property `Ellipse.minor_radius`: `pint.Quantity`

Minor radius of the ellipse.

property `Ellipse.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the ellipse.

property `Ellipse.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the ellipse.

property `Ellipse.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the ellipse.

property `Ellipse.eccentricity`: `ansys.geometry.core.typing.Real`

Eccentricity of the ellipse.

property `Ellipse.linear_eccentricity`: `pint.Quantity`

Linear eccentricity of the ellipse.

Notes

The linear eccentricity is the distance from the center to the focus.

property `Ellipse.semi_latus_rectum`: `pint.Quantity`

Semi-latus rectum of the ellipse.

property `Ellipse.perimeter`: `pint.Quantity`

Perimeter of the ellipse.

property `Ellipse.area`: `pint.Quantity`

Area of the ellipse.

Method detail

`Ellipse.__eq__(other: Ellipse) → bool`

Equals operator for the `Ellipse` class.

`Ellipse.mirrored_copy() → Ellipse`

Create a mirrored copy of the ellipse along the y-axis.

Returns

`Ellipse`

New ellipse that is a mirrored copy of the original ellipse.

`Ellipse.evaluate(parameter: ansys.geometry.core.typing.Real) → EllipseEvaluation`

Evaluate the ellipse at the given parameter.

Parameters

`parameter`

[Real] Parameter to evaluate the ellipse at.

Returns**EllipseEvaluation**

Resulting evaluation.

Ellipse.project_point(*point*: ansys.geometry.core.math.point.Point3D) → *EllipseEvaluation*

Project a point onto the ellipse and evaluate the ellipse.

Parameters**point**

[Point3D] Point to project onto the ellipse.

Returns**EllipseEvaluation**

Resulting evaluation.

Ellipse.is_coincident_ellipse(*other*: Ellipse) → **bool**

Determine if this ellipse is coincident with another.

Parameters**other**

[Ellipse] Ellipse to determine coincidence with.

Returns**bool**

True if this ellipse is coincident with the other, False otherwise.

Ellipse.transformed_copy(*matrix*: ansys.geometry.core.math.matrix.Matrix44) → *Ellipse*

Create a transformed copy of the ellipse from a transformation matrix.

Parameters**matrix**

[Matrix44] 4x4 transformation matrix to apply to the ellipse.

Returns**Ellipse**

New ellipse that is the transformed copy of the original ellipse.

Ellipse.parameterization() → *ansys.geometry.core.shapes.parameterization.Parameterization*

Get the parametrization of the ellipse.

The parameter of an ellipse specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .**Returns****Parameterization**

Information about how the ellipse is parameterized.

abstract Ellipse.contains_param(*param*: ansys.geometry.core.typing.Real) → **bool**

Check a parameter is within the parametric range of the curve.

abstract Ellipse.contains_point(*point*: ansys.geometry.core.math.point.Point3D) → **bool**

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

EllipseEvaluation

```
class ansys.geometry.core.shapes.curves.ellipse.EllipseEvaluation(ellipse: Ellipse, parameter:
                                                                    an-
                                                                    sys.geometry.core.typing.Real)
```

Bases: `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate an ellipse at a given parameter.

Parameters

ellipse: `~ansys.geometry.core.shapes.curves.ellipse.Ellipse`

Ellipse to evaluate.

parameter: `float, int`

Parameter to evaluate the ellipse at.

Overview

Properties

<i>ellipse</i>	Ellipse being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation.
<i>normal</i>	Normal of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the ellipse.

Import detail

```
from ansys.geometry.core.shapes.curves.ellipse import EllipseEvaluation
```

Property detail

property `EllipseEvaluation.ellipse:` *Ellipse*

Ellipse being evaluated.

property `EllipseEvaluation.parameter:` `ansys.geometry.core.typing.Real`

Parameter that the evaluation is based upon.

property `EllipseEvaluation.position:` `ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

Point3D

Point that lies on the ellipse at this evaluation.

property `EllipseEvaluation.tangent:` `ansys.geometry.core.math.vector.UnitVector3D`

Tangent of the evaluation.

Returns

UnitVector3D

Tangent unit vector to the ellipse at this evaluation.

property `EllipseEvaluation.normal`: `ansys.geometry.core.math.vector.UnitVector3D`

Normal of the evaluation.

Returns

UnitVector3D

Normal unit vector to the ellipse at this evaluation.

property `EllipseEvaluation.first_derivative`: `ansys.geometry.core.math.vector.Vector3D`

First derivative of the evaluation.

The first derivative is in the direction of the tangent and has a magnitude equal to the velocity (rate of change of position) at that point.

Returns

Vector3D

First derivative of the evaluation.

property `EllipseEvaluation.second_derivative`: `ansys.geometry.core.math.vector.Vector3D`

Second derivative of the evaluation.

Returns

Vector3D

Second derivative of the evaluation.

property `EllipseEvaluation.curvature`: `ansys.geometry.core.typing.Real`

Curvature of the ellipse.

Returns

Real

Curvature of the ellipse.

Description

Provides for creating and managing an ellipse.

The `line.py` module

Summary

Classes

<code>Line</code>	Provides 3D line representation.
<code>LineEvaluation</code>	Provides for evaluating a line.

Line

class `ansys.geometry.core.shapes.curves.line.Line`(*origin*: `numpy.ndarray` | `ansys.geometry.core.typing.RealSequence` | `ansys.geometry.core.math.point.Point3D`, *direction*: `numpy.ndarray` | `ansys.geometry.core.typing.RealSequence` | `ansys.geometry.core.math.vector.UnitVector3D` | `ansys.geometry.core.math.vector.Vector3D`)

Bases: `ansys.geometry.core.shapes.curves.curve.Curve`

Provides 3D line representation.

Parameters

origin

[`ndarray` | `RealSequence` | `Point3D`] Origin of the line.

direction

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] Direction of the line.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the curve.
<code>contains_point</code>	Check a point is contained by the curve.

Methods

<code>evaluate</code>	Evaluate the line at a given parameter.
<code>transformed_copy</code>	Create a transformed copy of the line from a transformation matrix.
<code>project_point</code>	Project a point onto the line and evaluate the line.
<code>is_coincident_line</code>	Determine if the line is coincident with another line.
<code>is_opposite_line</code>	Determine if the line is opposite another line.
<code>parameterization</code>	Get the parametrization of the line.

Properties

<code>origin</code>	Origin of the line.
<code>direction</code>	Direction of the line.

Special methods

<code>__eq__</code>	Equals operator for the Line class.
---------------------	-------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.line import Line
```

Property detail

property `Line.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the line.

property `Line.direction`: `ansys.geometry.core.math.vector.UnitVector3D`

Direction of the line.

Method detail

`Line.__eq__(other: object) → bool`

Equals operator for the Line class.

`Line.evaluate(parameter: float) → LineEvaluation`

Evaluate the line at a given parameter.

Parameters**parameter**

[Real] Parameter to evaluate the line at.

Returns**LineEvaluation**

Resulting evaluation.

`Line.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Line`

Create a transformed copy of the line from a transformation matrix.

Parameters**matrix**

[Matrix44] 4X4 transformation matrix to apply to the line.

Returns**Line**

New line that is the transformed copy of the original line.

`Line.project_point(point: ansys.geometry.core.math.point.Point3D) → LineEvaluation`

Project a point onto the line and evaluate the line.

Parameters**point**

[Point3D] Point to project onto the line.

Returns**LineEvaluation**

Resulting evaluation.

`Line.is_coincident_line(other: Line) → bool`

Determine if the line is coincident with another line.

Parameters**other**

[Line] Line to determine coincidence with.

Returns**bool**

True if the line is coincident with another line, False otherwise.

`Line.is_opposite_line(other: Line) → bool`

Determine if the line is opposite another line.

Parameters**other**

[Line] Line to determine opposition with.

Returns**bool**

True if the line is opposite to another line.

Line.parameterization() → *ansys.geometry.core.shapes.parameterization.Parameterization*

Get the parametrization of the line.

The parameter of a line specifies the distance from the *origin* in the direction of *direction*.**Returns****Parameterization**

Information about how the line is parameterized.

abstract Line.contains_param(*param: ansys.geometry.core.typing.Real*) → **bool**

Check a parameter is within the parametric range of the curve.

abstract Line.contains_point(*point: ansys.geometry.core.math.point.Point3D*) → **bool**

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

LineEvaluation**class** *ansys.geometry.core.shapes.curves.line.LineEvaluation*(*line: Line, parameter: float = None*)Bases: *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Provides for evaluating a line.

Overview**Properties**

<i>line</i>	Line being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the line, which is always 0.

Import detail

```
from ansys.geometry.core.shapes.curves.line import LineEvaluation
```

Property detail**property** *LineEvaluation.line: Line*

Line being evaluated.

property *LineEvaluation.parameter: float*

Parameter that the evaluation is based upon.

property `LineEvaluation.position`: *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the line at this evaluation.

property `LineEvaluation.tangent`: *ansys.geometry.core.math.vector.UnitVector3D*

Tangent of the evaluation.

Returns

UnitVector3D

Tangent unit vector to the line at this evaluation.

Notes

This is always equal to the direction of the line.

property `LineEvaluation.first_derivative`: *ansys.geometry.core.math.vector.Vector3D*

First derivative of the evaluation.

The first derivative is always equal to the direction of the line.

Returns

Vector3D

First derivative of the evaluation.

property `LineEvaluation.second_derivative`: *ansys.geometry.core.math.vector.Vector3D*

Second derivative of the evaluation.

The second derivative is always equal to a zero vector `Vector3D([0, 0, 0])`.

Returns

Vector3D

Second derivative of the evaluation, which is always `Vector3D([0, 0, 0])`.

property `LineEvaluation.curvature`: `float`

Curvature of the line, which is always 0.

Returns

Real

Curvature of the line, which is always 0.

Description

Provides for creating and managing a line.

The nurbs.py module

Summary

Classes

<i>NURBSCurve</i>	Represents a NURBS curve.
<i>NURBSCurveEvaluation</i>	Provides evaluation of a NURBS curve at a given parameter.

NURBSCurve

class `ansys.geometry.core.shapes.curves.nurbs.NURBSCurve`

Bases: `ansys.geometry.core.shapes.curves.curve.Curve`

Represents a NURBS curve.

Notes

This class is a wrapper around the NURBS curve class from the `geomdl` library. By leveraging the `geomdl` library, this class provides a high-level interface to create and manipulate NURBS curves. The `geomdl` library is a powerful library for working with NURBS curves and surfaces. For more information, see <https://pypi.org/project/geomdl/>.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the curve.
<code>contains_point</code>	Check a point is contained by the curve.

Constructors

<code>from_control_points</code>	Create a NURBS curve from control points.
----------------------------------	---

Methods

<code>parameterization</code>	Get the parametrization of the NURBS curve.
<code>transformed_copy</code>	Create a transformed copy of the curve.
<code>evaluate</code>	Evaluate the curve at the given parameter.
<code>project_point</code>	Project a point to the NURBS curve.

Properties

<code>geomdl_nurbs_curve</code>	Get the underlying NURBS curve.
<code>control_points</code>	Get the control points of the curve.
<code>degree</code>	Get the degree of the curve.
<code>knots</code>	Get the knot vector of the curve.
<code>weights</code>	Get the weights of the control points.

Special methods

<code>__eq__</code>	Determine if two curves are equal.
---------------------	------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.nurbs import NURBSCurve
```

Property detail

property `NURBSCurve.geomdl_nurbs_curve`: `geomdl.NURBS.Curve`

Get the underlying NURBS curve.

Notes

This property gives access to the full functionality of the NURBS curve coming from the *geomdl* library. Use with caution.

property `NURBSCurve.control_points`: `list[ansys.geometry.core.math.Point3D]`

Get the control points of the curve.

property `NURBSCurve.degree`: `int`

Get the degree of the curve.

property `NURBSCurve.knots`: `list[ansys.geometry.core.typing.Real]`

Get the knot vector of the curve.

property `NURBSCurve.weights`: `list[ansys.geometry.core.typing.Real]`

Get the weights of the control points.

Method detail

classmethod `NURBSCurve.from_control_points`(*control_points*: `list[ansys.geometry.core.math.Point3D]`,
degree: `int`, *knots*: `list[ansys.geometry.core.typing.Real]`,
weights: `list[ansys.geometry.core.typing.Real] = None`) → `NURBSCurve`

Create a NURBS curve from control points.

Parameters

control_points

`[list[Point3D]]` Control points of the curve.

degree

`[int]` Degree of the curve.

knots

`[list[Real]]` Knot vector of the curve.

weights

`[list[Real], optional]` Weights of the control points.

Returns

NURBSCurve

NURBS curve.

`NURBSCurve.__eq__`(*other*: `NURBSCurve`) → `bool`

Determine if two curves are equal.

`NURBSCurve.parameterization`() → `ansys.geometry.core.shapes.parameterization.Parameterization`

Get the parametrization of the NURBS curve.

The parameter is defined in the interval [0, 1] by default. Information is provided about the parameter type and form.

Returns

Parameterization

Information about how the NURBS curve is parameterized.

`NURBSCurve.transformed_copy(matrix: ansys.geometry.core.math.Matrix44) → NURBSCurve`

Create a transformed copy of the curve.

Parameters**matrix**

[Matrix44] Transformation matrix.

Returns**NURBSCurve**

Transformed copy of the curve.

`NURBSCurve.evaluate(parameter: ansys.geometry.core.typing.Real) →
ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate the curve at the given parameter.

Parameters**parameter**

[Real] Parameter to evaluate the curve at.

Returns**CurveEvaluation**

Evaluation of the curve at the given parameter.

`abstract NURBSCurve.contains_param(param: ansys.geometry.core.typing.Real) → bool`

Check a parameter is within the parametric range of the curve.

`abstract NURBSCurve.contains_point(point: ansys.geometry.core.math.Point3D) → bool`

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

`NURBSCurve.project_point(point: ansys.geometry.core.math.Point3D, initial_guess:
ansys.geometry.core.typing.Real | None = None) →
ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Project a point to the NURBS curve.

This method returns the evaluation at the closest point.

Parameters**point**

[Point3D] Point to project to the curve.

initial_guess

[Real, optional] Initial guess for the optimization algorithm. If not provided, the midpoint of the domain is used.

Returns**CurveEvaluation**

Evaluation at the closest point on the curve.

Notes

Based on [the NURBS book](#), the projection of a point to a NURBS curve is the solution to the following optimization problem: minimize the distance between the point and the curve. The distance is defined as the Euclidean distance squared. For more information, please refer to the implementation of the `distance_squared` function.

NURBSCurveEvaluation

```
class ansys.geometry.core.shapes.curves.nurbs.NURBSCurveEvaluation(nurbs_curve: NURBSCurve,
                                                                    parameter: an-
                                                                    sys.geometry.core.typing.Real)
```

Bases: `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Provides evaluation of a NURBS curve at a given parameter.

Parameters

nurbs_curve: `~ansys.geometry.core.shapes.curves.nurbs.NURBSCurve`
NURBS curve to evaluate.

parameter: `Real`
Parameter to evaluate the NURBS curve at.

Overview

Properties

<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the evaluation.

Import detail

```
from ansys.geometry.core.shapes.curves.nurbs import NURBSCurveEvaluation
```

Property detail

property `NURBSCurveEvaluation.parameter: ansys.geometry.core.typing.Real`
Parameter that the evaluation is based upon.

property `NURBSCurveEvaluation.position: ansys.geometry.core.math.Point3D`
Position of the evaluation.

property `NURBSCurveEvaluation.first_derivative: ansys.geometry.core.math.vector.Vector3D`
First derivative of the evaluation.

property `NURBSCurveEvaluation.second_derivative: ansys.geometry.core.math.vector.Vector3D`
Second derivative of the evaluation.

property `NURBSCurveEvaluation.curvature: ansys.geometry.core.typing.Real`
Curvature of the evaluation.

Description

Provides for creating and managing a NURBS curve.

The `trimmed_curve.py` module

Summary

Classes

<code>TrimmedCurve</code>	Represents a trimmed curve.
<code>ReversedTrimmedCurve</code>	Represents a reversed trimmed curve.

`TrimmedCurve`

```
class ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve(geometry: an-
    sys.geometry.core.shapes.curves.curve.Curve,
    start: an-
    sys.geometry.core.math.point.Point3D,
    end: an-
    sys.geometry.core.math.point.Point3D,
    interval: an-
    sys.geometry.core.shapes.parameterization.Interval,
    length: pint.Quantity,
    grpc_client: an-
    sys.geometry.core.connection.client.GrpcClient
    = None)
```

Represents a trimmed curve.

A trimmed curve is a curve that has a boundary. This boundary comes in the form of an interval.

Parameters

- `geometry`**
[Curve] Underlying mathematical representation of the curve.
- `start`**
[Point3D] Start point of the curve.
- `end`**
[Point3D] End point of the curve.
- `interval`**
[Interval] Parametric interval that bounds the curve.
- `length`**
[Quantity] Length of the curve.
- `grpc_client`**
[GrpcClient, default: `None`] gRPC client that is required for advanced functions such as `intersect_curve()`.

Overview

Methods

<code>evaluate_proportion</code>	Evaluate the curve at a proportional value.
<code>intersect_curve</code>	Get the intersect points of this trimmed curve with another one.

Properties

<code>geometry</code>	Underlying mathematical curve.
<code>start</code>	Start point of the curve.
<code>end</code>	End point of the curve.
<code>length</code>	Calculated length of the edge.
<code>interval</code>	Interval of the curve that provides its boundary.

Special methods

<code>__repr__</code>	Represent the trimmed curve as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.shapes.curves.trimmed_curve import TrimmedCurve
```

Property detail

property `TrimmedCurve.geometry`: `ansys.geometry.core.shapes.curves.curve.Curve`
Underlying mathematical curve.

property `TrimmedCurve.start`: `ansys.geometry.core.math.point.Point3D`
Start point of the curve.

property `TrimmedCurve.end`: `ansys.geometry.core.math.point.Point3D`
End point of the curve.

property `TrimmedCurve.length`: `pint.Quantity`
Calculated length of the edge.

property `TrimmedCurve.interval`: `ansys.geometry.core.shapes.parameterization.Interval`
Interval of the curve that provides its boundary.

Method detail

`TrimmedCurve.evaluate_proportion`(*param*: `ansys.geometry.core.typing.Real`) → `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate the curve at a proportional value.

A parameter of 0 corresponds to the start of the curve, while a parameter of 1 corresponds to the end of the curve.

Parameters

param

[Real] Parameter in the proportional range [0,1].

Returns

CurveEvaluation

Resulting curve evaluation.

TrimmedCurve.**intersect_curve**(*other*: TrimmedCurve) → list[ansys.geometry.core.math.point.Point3D]

Get the intersect points of this trimmed curve with another one.

If the two trimmed curves do not intersect, an empty list is returned.

Parameters

other

[TrimmedCurve] Trimmed curve to intersect with.

Returns

list[Point3D]

All points of intersection between the curves.

TrimmedCurve.**__repr__**() → str

Represent the trimmed curve as a string.

ReversedTrimmedCurve

```
class ansys.geometry.core.shapes.curves.trimmed_curve.ReversedTrimmedCurve(geometry: an-
    sys.geometry.core.shapes.curves.curv
    start: an-
    sys.geometry.core.math.point.Point3D
    end: an-
    sys.geometry.core.math.point.Point3D
    interval: an-
    sys.geometry.core.shapes.parameteriz
    length:
    pint.Quantity,
    grpc_client: an-
    sys.geometry.core.connection.client.C
    = None)
```

Bases: TrimmedCurve

Represents a reversed trimmed curve.

When a curve is reversed, its start and end points are swapped, and parameters for evaluations are handled to provide expected results conforming to the direction of the curve. For example, evaluating a trimmed curve proportionally at 0 evaluates at the start point of the curve, but evaluating a reversed trimmed curve proportionally at 0 evaluates at what was previously the end point of the curve but is now the start point.

Parameters

geometry

[Curve] Underlying mathematical representation of the curve.

start

[Point3D] Original start point of the curve.

end

[Point3D] Original end point of the curve.

interval

[Interval] Parametric interval that bounds the curve.

length

[Quantity] Length of the curve.

grpc_client

[GrpcClient, default: None] gRPC client that is required for advanced functions such as *intersect_curve()*.

Overview

Methods

evaluate_proportion Evaluate the curve at a proportional value.

Import detail

```
from ansys.geometry.core.shapes.curves.trimmed_curve import ReversedTrimmedCurve
```

Method detail

ReversedTrimmedCurve.**evaluate_proportion**(*param: ansys.geometry.core.typing.Real*) → *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Evaluate the curve at a proportional value.

A parameter of 0 corresponds to the start of the curve, while a parameter of 1 corresponds to the end of the curve.

Parameters

param

[Real] Parameter in the proportional range [0,1].

Returns

CurveEvaluation

Resulting curve evaluation.

Description

Trimmed curve class.

Description

Provides the PyAnsys Geometry curves subpackage.

The surfaces package

Summary

Submodules

<i>cone</i>	Provides for creating and managing a cone.
<i>cylinder</i>	Provides for creating and managing a cylinder.
<i>plane</i>	Provides for creating and managing a plane.
<i>sphere</i>	Provides for creating and managing a sphere.
<i>surface</i>	Provides the Surface class.
<i>surface_evaluation</i>	Provides for evaluating a surface.
<i>torus</i>	Provides for creating and managing a torus.
<i>trimmed_surface</i>	Provides the TrimmedSurface class.

The cone.py module

Summary

Classes

<i>Cone</i>	Provides 3D cone representation.
<i>ConeEvaluation</i>	Evaluate the cone at given parameters.

Cone

```
class ansys.geometry.core.shapes-surfaces.cone.Cone(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D, radius:
    pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance |
    ansys.geometry.core.typing.Real, half_angle:
    pint.Quantity |
    ansys.geometry.core.misc.measurements.Angle |
    ansys.geometry.core.typing.Real, reference:
    numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_X, axis: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes-surfaces.surface.Surface*

Provides 3D cone representation.

Parameters

origin

[*ndarray* | *RealSequence* | *Point3D*] Origin of the cone.

radius

[*Quantity* | *Distance* | *Real*] Radius of the cone.

half_angle

[*Quantity* | *Angle* | *Real*] Half angle of the apex, determining the upward angle.

reference

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] X-axis direction.

axis

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] Z-axis direction.

Overview**Abstract methods**

<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.

Methods

<i>transformed_copy</i>	Create a transformed copy of the cone from a transformation matrix.
<i>mirrored_copy</i>	Create a mirrored copy of the cone along the y-axis.
<i>evaluate</i>	Evaluate the cone at given parameters.
<i>project_point</i>	Project a point onto the cone and evaluate the cone.
<i>parameterization</i>	Parameterize the cone surface as a tuple (U and V respectively).

Properties

<i>origin</i>	Origin of the cone.
<i>radius</i>	Radius of the cone.
<i>half_angle</i>	Half angle of the apex.
<i>dir_x</i>	X-direction of the cone.
<i>dir_y</i>	Y-direction of the cone.
<i>dir_z</i>	Z-direction of the cone.
<i>height</i>	Height of the cone.
<i>surface_area</i>	Surface area of the cone.
<i>volume</i>	Volume of the cone.
<i>apex</i>	Apex point of the cone.
<i>apex_param</i>	Apex parameter of the cone.

Special methods

<code>__eq__</code>	Equals operator for the Cone class.
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Import detail

```
from ansys.geometry.core.shapes-surfaces.cone import Cone
```

Property detail

property Cone.**origin**: *ansys.geometry.core.math.point.Point3D*

Origin of the cone.

property `Cone.radius`: `pint.Quantity`

Radius of the cone.

property `Cone.half_angle`: `pint.Quantity`

Half angle of the apex.

property `Cone.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the cone.

property `Cone.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the cone.

property `Cone.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the cone.

property `Cone.height`: `pint.Quantity`

Height of the cone.

property `Cone.surface_area`: `pint.Quantity`

Surface area of the cone.

property `Cone.volume`: `pint.Quantity`

Volume of the cone.

property `Cone.apex`: `ansys.geometry.core.math.point.Point3D`

Apex point of the cone.

property `Cone.apex_param`: `ansys.geometry.core.typing.Real`

Apex parameter of the cone.

Method detail

`Cone.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Cone`

Create a transformed copy of the cone from a transformation matrix.

Parameters

`matrix`

[Matrix44] 4x4 transformation matrix to apply to the cone.

Returns

`Cone`

New cone that is the transformed copy of the original cone.

`Cone.mirrored_copy() → Cone`

Create a mirrored copy of the cone along the y-axis.

Returns

`Cone`

New cone that is a mirrored copy of the original cone.

`Cone.__eq__(other: Cone) → bool`

Equals operator for the Cone class.

`Cone.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → ConeEvaluation`

Evaluate the cone at given parameters.

Parameters

parameter

[ParamUV] Parameters (u,v) to evaluate the cone at.

Returns**ConeEvaluation**

Resulting evaluation.

Cone.**project_point**(*point*: ansys.geometry.core.math.point.Point3D) → *ConeEvaluation*

Project a point onto the cone and evaluate the cone.

Parameters**point**

[Point3D] Point to project onto the cone.

Returns**ConeEvaluation**

Resulting evaluation.

Cone.**parameterization**() → *tuple*[*ansys.geometry.core.shapes.parameterization.Parameterization*,
ansys.geometry.core.shapes.parameterization.Parameterization]

Parameterize the cone surface as a tuple (U and V respectively).

The U parameter specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the distance along the axis, with a zero parameter at the XY plane of the cone.

Returns**tuple[Parameterization, Parameterization]**

Information about how a cone's u and v parameters are parameterized, respectively.

abstract Cone.**contains_param**(*param_uv*: ansys.geometry.core.shapes.parameterization.ParamUV) → *bool*

Check a parameter is within the parametric range of the surface.

abstract Cone.**contains_point**(*point*: ansys.geometry.core.math.point.Point3D) → *bool*

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

ConeEvaluation

class ansys.geometry.core.shapes-surfaces-cone.ConeEvaluation(*cone*: Cone, *parameter*: ansys.geometry.core.shapes.parameterization.ParamUV)

Bases: *ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation*

Evaluate the cone at given parameters.

Parameters

cone: ~ansys.geometry.core.shapes-surfaces-cone.Cone

Cone to evaluate.

parameter: ParamUV

Pparameters (u, v) to evaluate the cone at.

Overview

Properties

<i>cone</i>	Cone being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	Second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	Second derivative with respect to the V parameter.
<i>min_curvature</i>	Minimum curvature of the cone.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature of the cone.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.cone import ConeEvaluation
```

Property detail

property `ConeEvaluation.cone`: **Cone**

Cone being evaluated.

property `ConeEvaluation.parameter`: ***ansys.geometry.core.shapes.parameterization.ParamUV***

Parameter that the evaluation is based upon.

property `ConeEvaluation.position`: ***ansys.geometry.core.math.point.Point3D***

Position of the evaluation.

Returns

Point3D

Point that lies on the cone at this evaluation.

property `ConeEvaluation.normal`: ***ansys.geometry.core.math.vector.UnitVector3D***

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the cone at this evaluation.

property `ConeEvaluation.u_derivative`: ***ansys.geometry.core.math.vector.Vector3D***

First derivative with respect to the U parameter.

Returns

Vector3D

First derivative with respect to the U parameter.

property `ConeEvaluation.v_derivative`: ***ansys.geometry.core.math.vector.Vector3D***

First derivative with respect to the V parameter.

Returns**Vector3D**

First derivative with respect to the V parameter.

property ConeEvaluation.uu_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U parameter.

Returns**Vector3D**

Second derivative with respect to the U parameter.

property ConeEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U and V parameters.

Returns**Vector3D**

Second derivative with respect to U and V parameters.

property ConeEvaluation.vv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the V parameter.

Returns**Vector3D**

Second derivative with respect to the V parameter.

property ConeEvaluation.min_curvature: *ansys.geometry.core.typing.Real*

Minimum curvature of the cone.

Returns**Real**

Minimum curvature of the cone.

property ConeEvaluation.min_curvature_direction:

ansys.geometry.core.math.vector.UnitVector3D

Minimum curvature direction.

Returns**UnitVector3D**

Minimum curvature direction.

property ConeEvaluation.max_curvature: *ansys.geometry.core.typing.Real*

Maximum curvature of the cone.

Returns**Real**

Maximum curvature of the cone.

property ConeEvaluation.max_curvature_direction:

ansys.geometry.core.math.vector.UnitVector3D

Maximum curvature direction.

Returns**UnitVector3D**

Maximum curvature direction.

Description

Provides for creating and managing a cone.

The `cylinder.py` module

Summary

Classes

<code>Cylinder</code>	Provides 3D cylinder representation.
<code>CylinderEvaluation</code>	Provides evaluation of a cylinder at given parameters.

Cylinder

```
class ansys.geometry.core.shapes_surfaces_cylinder.Cylinder(origin: numpy.ndarray | an-  
sys.geometry.core.typing.RealSequence  
| an-  
sys.geometry.core.math.point.Point3D,  
radius: pint.Quantity | an-  
sys.geometry.core.misc.measurements.Distance  
| ansys.geometry.core.typing.Real,  
reference: numpy.ndarray | an-  
sys.geometry.core.typing.RealSequence  
| an-  
sys.geometry.core.math.vector.UnitVector3D  
| an-  
sys.geometry.core.math.vector.Vector3D  
= UNITVECTOR3D_X, axis:  
numpy.ndarray | an-  
sys.geometry.core.typing.RealSequence  
| an-  
sys.geometry.core.math.vector.UnitVector3D  
| an-  
sys.geometry.core.math.vector.Vector3D  
= UNITVECTOR3D_Z)
```

Bases: `ansys.geometry.core.shapes_surfaces_surface.Surface`

Provides 3D cylinder representation.

Parameters

origin

[`ndarray` | `RealSequence` | `Point3D`] Origin of the cylinder.

radius

[`Quantity` | `Distance` | `Real`] Radius of the cylinder.

reference

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] X-axis direction.

axis

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the surface.
<code>contains_point</code>	Check a point is contained by the surface.

Methods

<code>surface_area</code>	Get the surface area of the cylinder.
<code>volume</code>	Get the volume of the cylinder.
<code>transformed_copy</code>	Create a transformed copy of the cylinder from a transformation matrix.
<code>mirrored_copy</code>	Create a mirrored copy of the cylinder along the y-axis.
<code>evaluate</code>	Evaluate the cylinder at the given parameters.
<code>project_point</code>	Project a point onto the cylinder and evaluate the cylinder.
<code>parameterization</code>	Parameterize the cylinder surface as a tuple (U and V respectively).

Properties

<code>origin</code>	Origin of the cylinder.
<code>radius</code>	Radius of the cylinder.
<code>dir_x</code>	X-direction of the cylinder.
<code>dir_y</code>	Y-direction of the cylinder.
<code>dir_z</code>	Z-direction of the cylinder.

Special methods

<code>__eq__</code>	Equals operator for the Cylinder class.
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Import detail

```
from ansys.geometry.core.shapes-surfaces.cylinder import Cylinder
```

Property detail

property `Cylinder.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the cylinder.

property `Cylinder.radius`: `pint.Quantity`

Radius of the cylinder.

property `Cylinder.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the cylinder.

property `Cylinder.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the cylinder.

property `Cylinder.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the cylinder.

Method detail

`Cylinder.surface_area`(*height*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*) → *pint.Quantity*

Get the surface area of the cylinder.

Parameters

height

[*Quantity* | *Distance* | *Real*] Height to bound the cylinder at.

Returns

Quantity

Surface area of the temporarily bounded cylinder.

Notes

By nature, a cylinder is infinite. If you want to get the surface area, you must bound it by a height. Normally a cylinder surface is not closed (does not have “caps” on the ends). This method assumes that the cylinder is closed for the purpose of getting the surface area.

`Cylinder.volume`(*height*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*) → *pint.Quantity*

Get the volume of the cylinder.

Parameters

height

[*Quantity* | *Distance* | *Real*] Height to bound the cylinder at.

Returns

Quantity

Volume of the temporarily bounded cylinder.

Notes

By nature, a cylinder is infinite. If you want to get the surface area, you must bound it by a height. Normally a cylinder surface is not closed (does not have “caps” on the ends). This method assumes that the cylinder is closed for the purpose of getting the surface area.

`Cylinder.transformed_copy`(*matrix*: *ansys.geometry.core.math.matrix.Matrix44*) → *Cylinder*

Create a transformed copy of the cylinder from a transformation matrix.

Parameters

matrix

[*Matrix44*] 4X4 transformation matrix to apply to the cylinder.

Returns

Cylinder

New cylinder that is the transformed copy of the original cylinder.

`Cylinder.mirrored_copy`() → *Cylinder*

Create a mirrored copy of the cylinder along the y-axis.

Returns

Cylinder

New cylinder that is a mirrored copy of the original cylinder.

`Cylinder.__eq__(other: Cylinder) → bool`

Equals operator for the Cylinder class.

`Cylinder.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → CylinderEvaluation`

Evaluate the cylinder at the given parameters.

Parameters

parameter

[ParamUV] Parameters (u,v) to evaluate the cylinder at.

Returns

CylinderEvaluation

Resulting evaluation.

`Cylinder.project_point(point: ansys.geometry.core.math.point.Point3D) → CylinderEvaluation`

Project a point onto the cylinder and evaluate the cylinder.

Parameters

point

[Point3D] Point to project onto the cylinder.

Returns

CylinderEvaluation

Resulting evaluation.

`Cylinder.parameterization() → tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the cylinder surface as a tuple (U and V respectively).

The U parameter specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the distance along the axis, with a zero parameter at the XY plane of the cylinder.

Returns

tuple[Parameterization, Parameterization]

Information about how a cylinder's u and v parameters are parameterized, respectively.

abstract `Cylinder.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`

Check a parameter is within the parametric range of the surface.

abstract `Cylinder.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

CylinderEvaluation

class `ansys.geometry.core.shapes_surfaces.cylinder.CylinderEvaluation(cylinder: Cylinder, parameter: ansys.geometry.core.shapes.parameterization.ParamUV)`

Bases: `ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation`

Provides evaluation of a cylinder at given parameters.

Parameters

cylinder: `~ansys.geometry.core.shapes-surfaces.cylinder.Cylinder`
Cylinder to evaluate.

parameter: `ParamUV`
Parameters (u, v) to evaluate the cylinder at.

Overview

Properties

<code>cylinder</code>	Cylinder being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	Second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	Second derivative with respect to the V parameter.
<code>min_curvature</code>	Minimum curvature of the cylinder.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature of the cylinder.
<code>max_curvature_direction</code>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.cylinder import CylinderEvaluation
```

Property detail

property `CylinderEvaluation.cylinder:` *Cylinder*
Cylinder being evaluated.

property `CylinderEvaluation.parameter:`
ansys.geometry.core.shapes-parameterization.ParamUV
Parameter that the evaluation is based upon.

property `CylinderEvaluation.position:` *ansys.geometry.core.math.point.Point3D*
Position of the evaluation.

Returns

Point3D
Point that lies on the cylinder at this evaluation.

property `CylinderEvaluation.normal:` *ansys.geometry.core.math.vector.UnitVector3D*
Normal to the surface.

Returns

UnitVector3D
Normal unit vector to the cylinder at this evaluation.

property `CylinderEvaluation.u_derivative:` *ansys.geometry.core.math.vector.Vector3D*
First derivative with respect to the U parameter.

Returns**Vector3D**

First derivative with respect to the U parameter.

property `CylinderEvaluation.v_derivative`: *ansys.geometry.core.math.vector.Vector3D*
 First derivative with respect to the V parameter.

Returns**Vector3D**

First derivative with respect to the V parameter.

property `CylinderEvaluation.uu_derivative`: *ansys.geometry.core.math.vector.Vector3D*
 Second derivative with respect to the U parameter.

Returns**Vector3D**

Second derivative with respect to the U parameter.

property `CylinderEvaluation.uv_derivative`: *ansys.geometry.core.math.vector.Vector3D*
 Second derivative with respect to the U and V parameters.

Returns**Vector3D**

Second derivative with respect to the U and v parameters.

property `CylinderEvaluation.vv_derivative`: *ansys.geometry.core.math.vector.Vector3D*
 Second derivative with respect to the V parameter.

Returns**Vector3D**

Second derivative with respect to the V parameter.

property `CylinderEvaluation.min_curvature`: *ansys.geometry.core.typing.Real*
 Minimum curvature of the cylinder.

Returns**Real**

Minimum curvature of the cylinder.

property `CylinderEvaluation.min_curvature_direction`:
ansys.geometry.core.math.vector.UnitVector3D

Minimum curvature direction.

Returns**UnitVector3D**

Minimum curvature direction.

property `CylinderEvaluation.max_curvature`: *ansys.geometry.core.typing.Real*
 Maximum curvature of the cylinder.

Returns**Real**

Maximum curvature of the cylinder.

property `CylinderEvaluation.max_curvature_direction:`
`ansys.geometry.core.math.vector.UnitVector3D`

Maximum curvature direction.

Returns

UnitVector3D

Maximum curvature direction.

Description

Provides for creating and managing a cylinder.

The `plane.py` module

Summary

Classes

<code>PlaneSurface</code>	Provides 3D plane surface representation.
<code>PlaneEvaluation</code>	Provides evaluation of a plane at given parameters.

PlaneSurface

class `ansys.geometry.core.shapes-surfaces.plane.PlaneSurface`(*origin: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D, reference: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = UNITVECTOR3D_X, axis: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = UNITVECTOR3D_Z*)

Bases: `ansys.geometry.core.shapes-surfaces.surface.Surface`

Provides 3D plane surface representation.

Parameters

origin

[`ndarray` | `RealSequence` | `Point3D`] Centered origin of the plane.

reference

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] X-axis direction.

axis

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] X-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Check a ParamUV is within the parametric range of the surface.
<code>contains_point</code>	Check whether a 3D point is in the domain of the plane.

Methods

<code>parameterization</code>	Parametrize the plane.
<code>project_point</code>	Evaluate the plane at a given 3D point.
<code>transformed_copy</code>	Get a transformed version of the plane given the transform.
<code>evaluate</code>	Evaluate the plane at a given u and v parameter.

Properties

<code>origin</code>	Origin of the plane.
<code>dir_x</code>	X-direction of the plane.
<code>dir_y</code>	Y-direction of the plane.
<code>dir_z</code>	Z-direction of the plane.

Special methods

<code>__eq__</code>	Check whether two planes are equal.
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Import detail

```
from ansys.geometry.core.shapes-surfaces.plane import PlaneSurface
```

Property detail

property `PlaneSurface.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the plane.

property `PlaneSurface.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the plane.

property `PlaneSurface.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the plane.

property `PlaneSurface.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the plane.

Method detail

`PlaneSurface.__eq__(other: PlaneSurface) → bool`

Check whether two planes are equal.

abstract `PlaneSurface.contains_param`(*param_uv*: `ansys.geometry.core.shapes.parameterization.ParamUV`)
 → `bool`

Check a ParamUV is within the parametric range of the surface.

abstract `PlaneSurface.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Check whether a 3D point is in the domain of the plane.

`PlaneSurface.parameterization`() → `tuple`[`ansys.geometry.core.shapes.parameterization.Parameterization`,
 `ansys.geometry.core.shapes.parameterization.Parameterization`]

Parametrize the plane.

`PlaneSurface.project_point`(*point*: `ansys.geometry.core.math.point.Point3D`) →
 `ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation`

Evaluate the plane at a given 3D point.

`PlaneSurface.transformed_copy`(*matrix*: `ansys.geometry.core.math.matrix.Matrix44`) →
 `ansys.geometry.core.shapes_surfaces.surface.Surface`

Get a transformed version of the plane given the transform.

`PlaneSurface.evaluate`(*parameter*: `ansys.geometry.core.shapes.parameterization.ParamUV`) →
 `PlaneEvaluation`

Evaluate the plane at a given u and v parameter.

PlaneEvaluation

class `ansys.geometry.core.shapes_surfaces.plane.PlaneEvaluation`(*plane*: `PlaneSurface`, *parameter*:
 `ansys.geometry.core.shapes.parameterization.ParamUV`)

Bases: `ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation`

Provides evaluation of a plane at given parameters.

Parameters

plane: `~ansys.geometry.core.shapes_surfaces.plane.PlaneSurface`
 Plane to evaluate.

parameter: `ParamUV`
 Parameters (u, v) to evaluate the plane at.

Overview

Properties

<i>plane</i>	Plane being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Point on the surface, based on the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to u.
<i>v_derivative</i>	First derivative with respect to v.
<i>uu_derivative</i>	Second derivative with respect to u.
<i>uv_derivative</i>	Second derivative with respect to u and v.
<i>vv_derivative</i>	Second derivative with respect to v.
<i>min_curvature</i>	Minimum curvature.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.plane import PlaneEvaluation
```

Property detail

property `PlaneEvaluation.plane`: *PlaneSurface*

Plane being evaluated.

property `PlaneEvaluation.parameter`: *ansys.geometry.core.shapes.parameterization.ParamUV*

Parameter that the evaluation is based upon.

property `PlaneEvaluation.position`: *ansys.geometry.core.math.point.Point3D*

Point on the surface, based on the evaluation.

property `PlaneEvaluation.normal`: *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

property `PlaneEvaluation.u_derivative`: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to u.

property `PlaneEvaluation.v_derivative`: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to v.

property `PlaneEvaluation.uu_derivative`: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to u.

property `PlaneEvaluation.uv_derivative`: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to u and v.

property `PlaneEvaluation.vv_derivative`: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to v.

property `PlaneEvaluation.min_curvature`: *ansys.geometry.core.typing.Real*

Minimum curvature.

property `PlaneEvaluation.min_curvature_direction:`
`ansys.geometry.core.math.vector.UnitVector3D`

Minimum curvature direction.

property `PlaneEvaluation.max_curvature:` `ansys.geometry.core.typing.Real`

Maximum curvature.

property `PlaneEvaluation.max_curvature_direction:`
`ansys.geometry.core.math.vector.UnitVector3D`

Maximum curvature direction.

Description

Provides for creating and managing a plane.

The `sphere.py` module

Summary

Classes

<code>Sphere</code>	Provides 3D sphere representation.
<code>SphereEvaluation</code>	Evaluate a sphere at given parameters.

Sphere

```
class ansys.geometry.core.shapes.surfaces.sphere.Sphere(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D,
    radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real,
    reference: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D
    |
    ansys.geometry.core.math.vector.Vector3D
    = UNITVECTOR3D_X, axis:
    numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D
    |
    ansys.geometry.core.math.vector.Vector3D
    = UNITVECTOR3D_Z)
```

Bases: `ansys.geometry.core.shapes.surfaces.surface.Surface`

Provides 3D sphere representation.

Parameters

origin

`[ndarray | RealSequence | Point3D]` Origin of the sphere.

radius

[Quantity | Distance | Real] Radius of the sphere.

reference

[ndarray | RealSequence | UnitVector3D | Vector3D] X-axis direction.

axis

[ndarray | RealSequence | UnitVector3D | Vector3D] Z-axis direction.

Overview**Abstract methods**

<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.

Methods

<i>transformed_copy</i>	Create a transformed copy of the sphere from a transformation matrix.
<i>mirrored_copy</i>	Create a mirrored copy of the sphere along the y-axis.
<i>evaluate</i>	Evaluate the sphere at the given parameters.
<i>project_point</i>	Project a point onto the sphere and evaluate the sphere.
<i>parameterization</i>	Parameterization of the sphere surface as a tuple (U, V).

Properties

<i>origin</i>	Origin of the sphere.
<i>radius</i>	Radius of the sphere.
<i>dir_x</i>	X-direction of the sphere.
<i>dir_y</i>	Y-direction of the sphere.
<i>dir_z</i>	Z-direction of the sphere.
<i>surface_area</i>	Surface area of the sphere.
<i>volume</i>	Volume of the sphere.

Special methods

<code>__eq__</code>	Equals operator for the Sphere class.
---------------------	---------------------------------------

Import detail

```
from ansys.geometry.core.shapes.surfaces.sphere import Sphere
```

Property detail

property Sphere.*origin*: *ansys.geometry.core.math.point.Point3D*

Origin of the sphere.

property Sphere.*radius*: *pint.Quantity*

Radius of the sphere.

property Sphere.`dir_x`: *ansys.geometry.core.math.vector.UnitVector3D*

X-direction of the sphere.

property Sphere.`dir_y`: *ansys.geometry.core.math.vector.UnitVector3D*

Y-direction of the sphere.

property Sphere.`dir_z`: *ansys.geometry.core.math.vector.UnitVector3D*

Z-direction of the sphere.

property Sphere.`surface_area`: *pint.Quantity*

Surface area of the sphere.

property Sphere.`volume`: *pint.Quantity*

Volume of the sphere.

Method detail

Sphere.`__eq__`(*other*: Sphere) → bool

Equals operator for the Sphere class.

Sphere.`transformed_copy`(*matrix*: *ansys.geometry.core.math.matrix.Matrix44*) → *Sphere*

Create a transformed copy of the sphere from a transformation matrix.

Parameters

matrix

[*Matrix44*] 4X4 transformation matrix to apply to the sphere.

Returns

Sphere

New sphere that is the transformed copy of the original sphere.

Sphere.`mirrored_copy`() → *Sphere*

Create a mirrored copy of the sphere along the y-axis.

Returns

Sphere

New sphere that is a mirrored copy of the original sphere.

Sphere.`evaluate`(*parameter*: *ansys.geometry.core.shapes.parameterization.ParamUV*) → *SphereEvaluation*

Evaluate the sphere at the given parameters.

Parameters

parameter

[*ParamUV*] Parameters (u,v) to evaluate the sphere at.

Returns

SphereEvaluation

Resulting evaluation.

Sphere.`project_point`(*point*: *ansys.geometry.core.math.point.Point3D*) → *SphereEvaluation*

Project a point onto the sphere and evaluate the sphere.

Parameters

point

[*Point3D*] Point to project onto the sphere.

Returns**SphereEvaluation**

Resulting evaluation.

`Sphere.parameterization()` → `tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterization of the sphere surface as a tuple (U, V).

The U parameter specifies the longitude angle, increasing clockwise (east) about `dir_z` (right-hand corkscrew law). It has a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the latitude, increasing north, with a zero parameter at the equator and a range of $[-\pi/2, \pi/2]$.

Returns**tuple[Parameterization, Parameterization]**

Information about how a sphere's u and v parameters are parameterized, respectively.

abstract `Sphere.contains_param`(*param_uv*: `ansys.geometry.core.shapes.parameterization.ParamUV`) → `bool`

Check a parameter is within the parametric range of the surface.

abstract `Sphere.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

SphereEvaluation

class `ansys.geometry.core.shapes.surfaces.sphere.SphereEvaluation`(*sphere*: `Sphere`, *parameter*: `ansys.geometry.core.shapes.parameterization.ParamUV`)

Bases: `ansys.geometry.core.shapes.surfaces.surface_evaluation.SurfaceEvaluation`

Evaluate a sphere at given parameters.

Parameters

sphere: `~ansys.geometry.core.shapes.surfaces.sphere.Sphere`

Sphere to evaluate.

parameter: `ParamUV`

Parameters (u, v) to evaluate the sphere at.

Overview

Properties

<i>sphere</i>	Sphere being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>normal</i>	The normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	Second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	Second derivative with respect to the V parameter.
<i>min_curvature</i>	Minimum curvature of the sphere.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature of the sphere.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.sphere import SphereEvaluation
```

Property detail

property SphereEvaluation.sphere: *Sphere*

Sphere being evaluated.

property SphereEvaluation.parameter: *ansys.geometry.core.shapes.parameterization.ParamUV*

Parameter that the evaluation is based upon.

property SphereEvaluation.position: *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the sphere at this evaluation.

property SphereEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

The normal to the surface.

Returns

UnitVector3D

Normal unit vector to the sphere at this evaluation.

property SphereEvaluation.u_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the U parameter.

Returns

Vector3D

First derivative with respect to the U parameter.

property SphereEvaluation.v_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the V parameter.

Returns

Vector3D

First derivative with respect to the V parameter.

property SphereEvaluation.uu_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U parameter.

Returns**Vector3D**

Second derivative with respect to the U parameter.

property SphereEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U and V parameters.

Returns**Vector3D**

The second derivative with respect to the U and V parameters.

property SphereEvaluation.vv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the V parameter.

Returns**Vector3D**

The second derivative with respect to the V parameter.

property SphereEvaluation.min_curvature: *ansys.geometry.core.typing.Real*

Minimum curvature of the sphere.

Returns**Real**

Minimum curvature of the sphere.

property SphereEvaluation.min_curvature_direction:

ansys.geometry.core.math.vector.UnitVector3D

Minimum curvature direction.

Returns**UnitVector3D**

Minimum curvature direction.

property SphereEvaluation.max_curvature: *ansys.geometry.core.typing.Real*

Maximum curvature of the sphere.

Returns**Real**

Maximum curvature of the sphere.

property SphereEvaluation.max_curvature_direction:

ansys.geometry.core.math.vector.UnitVector3D

Maximum curvature direction.

Returns**UnitVector3D**

Maximum curvature direction.

Description

Provides for creating and managing a sphere.

The `surface.py` module

Summary

Classes

<i>Surface</i>	Provides the abstract base class for a 3D surface.
----------------	--

Surface

class `ansys.geometry.core.shapes.surfaces.surface.Surface`

Bases: `abc.ABC`

Provides the abstract base class for a 3D surface.

Overview

Abstract methods

<i>parameterization</i>	Parameterize the surface as a tuple (U and V respectively).
<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.
<i>transformed_copy</i>	Create a transformed copy of the surface.
<i>__eq__</i>	Determine if two surfaces are equal.
<i>evaluate</i>	Evaluate the surface at the given parameter.
<i>project_point</i>	Project a point to the surface.

Methods

<i>trim</i>	Trim this surface by bounding it with a BoxUV.
-------------	--

Import detail

```
from ansys.geometry.core.shapes.surfaces.surface import Surface
```

Method detail

abstract `Surface.parameterization()` →

`tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the surface as a tuple (U and V respectively).

abstract `Surface.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`

Check a parameter is within the parametric range of the surface.

abstract `Surface.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

abstract `Surface.transformed_copy`(*matrix*: `ansys.geometry.core.math.matrix.Matrix44`) → `Surface`

Create a transformed copy of the surface.

abstract `Surface.__eq__`(*other*: `Surface`) → `bool`

Determine if two surfaces are equal.

abstract `Surface.evaluate`(*parameter*: `ansys.geometry.core.shapes.parameterization.ParamUV`) → `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`

Evaluate the surface at the given parameter.

abstract `Surface.project_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`

Project a point to the surface.

This method returns the evaluation at the closest point.

`Surface.trim`(*box_uv*: `ansys.geometry.core.shapes-box_uv.BoxUV`) → `ansys.geometry.core.shapes-surfaces-trimmed_surface.TrimmedSurface`

Trim this surface by bounding it with a BoxUV.

Returns

TrimmedSurface

The resulting bounded surface.

Description

Provides the Surface class.

The surface_evaluation.py module

Summary

Classes

<code>SurfaceEvaluation</code>	Provides for evaluating a surface.
--------------------------------	------------------------------------

SurfaceEvaluation

class `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`(*parameter*: `ansys.geometry.core.shapes-parameterization.ParamUV`)

Provides for evaluating a surface.

Overview

Properties

<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Point on the surface, based on the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	The second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	The second derivative with respect to v.
<i>min_curvature</i>	Minimum curvature.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.surface_evaluation import SurfaceEvaluation
```

Property detail

property SurfaceEvaluation.**parameter**: *ansys.geometry.core.shapes.parameterization.ParamUV*

Abstractmethod

Parameter that the evaluation is based upon.

property SurfaceEvaluation.**position**: *ansys.geometry.core.math.point.Point3D*

Abstractmethod

Point on the surface, based on the evaluation.

property SurfaceEvaluation.**normal**: *ansys.geometry.core.math.vector.UnitVector3D*

Abstractmethod

Normal to the surface.

property SurfaceEvaluation.**u_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

First derivative with respect to the U parameter.

property SurfaceEvaluation.**v_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

First derivative with respect to the V parameter.

property SurfaceEvaluation.**uu_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

Second derivative with respect to the U parameter.

property SurfaceEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

The second derivative with respect to the U and V parameters.

property SurfaceEvaluation.vv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

The second derivative with respect to v.

property SurfaceEvaluation.min_curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Minimum curvature.

property SurfaceEvaluation.min_curvature_direction:
ansys.geometry.core.math.vector.UnitVector3D

Abstractmethod

Minimum curvature direction.

property SurfaceEvaluation.max_curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Maximum curvature.

property SurfaceEvaluation.max_curvature_direction:
ansys.geometry.core.math.vector.UnitVector3D

Abstractmethod

Maximum curvature direction.

Description

Provides for evaluating a surface.

The torus.py module

Summary

Classes

<i>Torus</i>	Provides 3D torus representation.
<i>TorusEvaluation</i>	Evaluate the torus`` at given parameters.

Torus

```
class ansys.geometry.core.shapes.surfaces.torus.Torus(origin: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.point.Point3D,
major_radius: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real,
minor_radius: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real, reference:
numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
an-
sys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_X, axis: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
an-
sys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes.surfaces.surface.Surface*

Provides 3D torus representation.

Parameters

origin

[*ndarray* | *RealSequence* | *Point3D*] Centered origin of the torus.

major_radius

[*Quantity* | *Distance* | *Real*] Major radius of the torus.

minor_radius

[*Quantity* | *Distance* | *Real*] Minor radius of the torus.

reference

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] X-axis direction.

axis

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] Z-axis direction.

Overview

Abstract methods

<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.

Methods

<i>transformed_copy</i>	Create a transformed copy of the torus from a transformation matrix.
<i>mirrored_copy</i>	Create a mirrored copy of the torus along the y-axis.
<i>evaluate</i>	Evaluate the torus at the given parameters.
<i>parameterization</i>	Parameterize the torus surface as a tuple (U and V respectively).
<i>project_point</i>	Project a point onto the torus and evaluate the torus.

Properties

<code>origin</code>	Origin of the torus.
<code>major_radius</code>	Semi-major radius of the torus.
<code>minor_radius</code>	Semi-minor radius of the torus.
<code>dir_x</code>	X-direction of the torus.
<code>dir_y</code>	Y-direction of the torus.
<code>dir_z</code>	Z-direction of the torus.
<code>volume</code>	Volume of the torus.
<code>surface_area</code>	Surface_area of the torus.

Special methods

<code>__eq__</code>	Equals operator for the Torus class.
---------------------	--------------------------------------

Import detail

```
from ansys.geometry.core.shapes-surfaces.torus import Torus
```

Property detail

property `Torus.origin`: `ansys.geometry.core.math.point.Point3D`
Origin of the torus.

property `Torus.major_radius`: `pint.Quantity`
Semi-major radius of the torus.

property `Torus.minor_radius`: `pint.Quantity`
Semi-minor radius of the torus.

property `Torus.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`
X-direction of the torus.

property `Torus.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`
Y-direction of the torus.

property `Torus.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`
Z-direction of the torus.

property `Torus.volume`: `pint.Quantity`
Volume of the torus.

property `Torus.surface_area`: `pint.Quantity`
Surface_area of the torus.

Method detail

`Torus.__eq__(other: Torus) → bool`
Equals operator for the Torus class.

`Torus.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Torus`
Create a transformed copy of the torus from a transformation matrix.

Parameters**matrix**

[Matrix44] 4x4 transformation matrix to apply to the torus.

Returns**Torus**

New torus that is the transformed copy of the original torus.

Torus.**mirrored_copy**() → *Torus*

Create a mirrored copy of the torus along the y-axis.

Returns**Torus**

New torus that is a mirrored copy of the original torus.

Torus.**evaluate**(*parameter*: ansys.geometry.core.shapes.parameterization.ParamUV) → *TorusEvaluation*

Evaluate the torus at the given parameters.

Parameters**parameter**

[ParamUV] Parameters (u,v) to evaluate the torus at.

Returns**TorusEvaluation**

Resulting evaluation.

Torus.**parameterization**() → tuple[*ansys.geometry.core.shapes.parameterization.Parameterization*,
ansys.geometry.core.shapes.parameterization.Parameterization]

Parameterize the torus surface as a tuple (U and V respectively).

The U parameter specifies the longitude angle, increasing clockwise (east) about the axis (right-hand corkscrew law). It has a zero parameter at `Geometry.Frame.DirX` and a period of 2π .

The V parameter specifies the latitude, increasing north, with a zero parameter at the equator. For the donut, where the major radius is greater than the minor radius, the range is $[-\pi, \pi]$ and the parameterization is periodic. For a degenerate torus, the range is restricted accordingly and the parameterization is non-periodic.

Returns**tuple[Parameterization, Parameterization]**

Information about how a torus's u and v parameters are parameterized, respectively.

Torus.**project_point**(*point*: ansys.geometry.core.math.point.Point3D) → *TorusEvaluation*

Project a point onto the torus and evaluate the torus.

Parameters**point**

[Point3D] Point to project onto the torus.

Returns**TorusEvaluation**

Resulting evaluation.

abstract Torus.**contains_param**(*param_uv*: ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a parameter is within the parametric range of the surface.

abstract `Torus.contains_point` (*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

TorusEvaluation

class `ansys.geometry.core.shapes-surfaces.torus.TorusEvaluation` (*torus*: `Torus`, *parameter*: `ansys.geometry.core.shapes.parameterization.ParamUV`)

Bases: `ansys.geometry.core.shapes-surfaces.surface_evaluation.SurfaceEvaluation`

Evaluate the torus at given parameters.

Parameters

Torus: `~ansys.geometry.core.shapes-surfaces.torus.Torus`

Torus to evaluate.

parameter: `ParamUV`

Parameters (u, v) to evaluate the torus at.

Overview

Properties

<code>torus</code>	Torus being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	Second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	Second derivative with respect to the V parameter.
<code>curvature</code>	Curvature of the torus.
<code>min_curvature</code>	Minimum curvature of the torus.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature of the torus.
<code>max_curvature_direction</code>	Maximum curvature direction.

Attributes

`cache`

Import detail

```
from ansys.geometry.core.shapes-surfaces.torus import TorusEvaluation
```

Property detail

property `TorusEvaluation.torus`: `Torus`

Torus being evaluated.

property `TorusEvaluation.parameter`: *ansys.geometry.core.shapes.parameterization.ParamUV*
Parameter that the evaluation is based upon.

property `TorusEvaluation.position`: *ansys.geometry.core.math.point.Point3D*
Position of the evaluation.

Returns

Point3D

Point that lies on the torus at this evaluation.

property `TorusEvaluation.normal`: *ansys.geometry.core.math.vector.UnitVector3D*
Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the torus at this evaluation.

property `TorusEvaluation.u_derivative`: *ansys.geometry.core.math.vector.Vector3D*
First derivative with respect to the U parameter.

Returns

Vector3D

First derivative with respect to the U parameter.

property `TorusEvaluation.v_derivative`: *ansys.geometry.core.math.vector.Vector3D*
First derivative with respect to the V parameter.

Returns

Vector3D

First derivative with respect to the V parameter.

property `TorusEvaluation.uu_derivative`: *ansys.geometry.core.math.vector.Vector3D*
Second derivative with respect to the U parameter.

Returns

Vector3D

Second derivative with respect to the U parameter.

property `TorusEvaluation.uv_derivative`: *ansys.geometry.core.math.vector.Vector3D*
Second derivative with respect to the U and V parameters.

Returns

Vector3D

Second derivative with respect to the U and V parameters.

property `TorusEvaluation.vv_derivative`: *ansys.geometry.core.math.vector.Vector3D*
Second derivative with respect to the V parameter.

Returns

Vector3D

Second derivative with respect to the V parameter.

property `TorusEvaluation.curvature`: `tuple`[*ansys.geometry.core.typing.Real*,
ansys.geometry.core.math.vector.Vector3D, *ansys.geometry.core.typing.Real*,
ansys.geometry.core.math.vector.Vector3D]

Curvature of the torus.

Returns**tuple**[Real, Vector3D, Real, Vector3D]

Minimum and maximum curvature value and direction, respectively.

property TorusEvaluation.**min_curvature**: **ansys.geometry.core.typing.Real**

Minimum curvature of the torus.

Returns**Real**

Minimum curvature of the torus.

property TorusEvaluation.**min_curvature_direction**:**ansys.geometry.core.math.vector.UnitVector3D**

Minimum curvature direction.

Returns**UnitVector3D**

Minimum curvature direction.

property TorusEvaluation.**max_curvature**: **ansys.geometry.core.typing.Real**

Maximum curvature of the torus.

Returns**Real**

Maximum curvature of the torus.

property TorusEvaluation.**max_curvature_direction**:**ansys.geometry.core.math.vector.UnitVector3D**

Maximum curvature direction.

Returns**UnitVector3D**

Maximum curvature direction.

Attribute detailTorusEvaluation.**cache****Description**

Provides for creating and managing a torus.

The trimmed_surface.py module**Summary****Classes**

<i>TrimmedSurface</i>	Represents a trimmed surface.
<i>ReversedTrimmedSurface</i>	Represents a reversed trimmed surface.

TrimmedSurface

```
class ansys.geometry.core.shapes_surfaces_trimmed_surface.TrimmedSurface(geometry: an-
                                                                    sys.geometry.core.shapes_surfaces_surfa
                                                                    box_uv: an-
                                                                    sys.geometry.core.shapes_box_uv.BoxU
```

Represents a trimmed surface.

A trimmed surface is a surface that has a boundary. This boundary comes in the form of a bounding BoxUV.

Parameters

face

[Face] Face that the trimmed surface belongs to.

geometry

[Surface] Underlying mathematical representation of the surface.

Overview

Methods

<i>get_proportional_parameters</i>	Convert non-proportional parameters into proportional parameters.
<i>normal</i>	Provide the normal to the surface.
<i>project_point</i>	Project a point onto the surface and evaluate it at that location.
<i>evaluate_proportion</i>	Evaluate the surface at proportional u and v parameters.

Properties

<i>geometry</i>	Underlying mathematical surface.
<i>box_uv</i>	Bounding BoxUV of the surface.

Import detail

```
from ansys.geometry.core.shapes_surfaces_trimmed_surface import TrimmedSurface
```

Property detail

property TrimmedSurface.**geometry**: *ansys.geometry.core.shapes_surfaces_surface.Surface*
Underlying mathematical surface.

property TrimmedSurface.**box_uv**: *ansys.geometry.core.shapes_box_uv.BoxUV*
Bounding BoxUV of the surface.

Method detail

TrimmedSurface.**get_proportional_parameters**(*param_uv*:
ansys.geometry.core.shapes_parameterization.ParamUV) →
ansys.geometry.core.shapes_parameterization.ParamUV

Convert non-proportional parameters into proportional parameters.

Parameters

param_uv
[ParamUV] Non-proportional UV parameters.

Returns

ParamUV
Proportional (from 0-1) UV parameters.

`TrimmedSurface.normal`(*u*: *ansys.geometry.core.typing.Real*, *v*: *ansys.geometry.core.typing.Real*) → *ansys.geometry.core.math.vector.UnitVector3D*

Provide the normal to the surface.

Parameters

u
[Real] First coordinate of the 2D representation of a surface in UV space.

v
[Real] Second coordinate of the 2D representation of a surface in UV space.

Returns

UnitVector3D
Unit vector is normal to the surface at the given UV coordinates.

`TrimmedSurface.project_point`(*point*: *ansys.geometry.core.math.point.Point3D*) → *ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation*

Project a point onto the surface and evaluate it at that location.

Parameters

point
[Point3D] Point to project onto the surface.

Returns

SurfaceEvaluation
Resulting evaluation.

`TrimmedSurface.evaluate_proportion`(*u*: *ansys.geometry.core.typing.Real*, *v*: *ansys.geometry.core.typing.Real*) → *ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation*

Evaluate the surface at proportional u and v parameters.

Parameters

u
[Real] U parameter in the proportional range [0,1].

v
[Real] V parameter in the proportional range [0,1].

Returns

SurfaceEvaluation
Resulting surface evaluation.

ReversedTrimmedSurface

```
class ansys.geometry.core.shapes_surfaces_trimmed_surface.ReversedTrimmedSurface(geometry:
    an-
    sys.geometry.core.shapes_surfaces_trimmed_surface.Box
    box_uv:
    an-
    sys.geometry.core.shapes_surfaces_trimmed_surface.Box
```

Bases: TrimmedSurface

Represents a reversed trimmed surface.

When a surface is reversed, its normal vector is negated to provide the proper outward facing vector.

Parameters

face

[Face] Face that the trimmed surface belongs to.

geometry

[Surface] Underlying mathematical representation of the surface.

Overview

Methods

<i>normal</i>	Provide the normal to the surface.
<i>project_point</i>	Project a point onto the surface and evaluate it at that location.

Import detail

```
from ansys.geometry.core.shapes_surfaces_trimmed_surface import ReversedTrimmedSurface
```

Method detail

`ReversedTrimmedSurface.normal`(*u*: *ansys.geometry.core.typing.Real*, *v*: *ansys.geometry.core.typing.Real*) → *ansys.geometry.core.math.vector.UnitVector3D*

Provide the normal to the surface.

Parameters

u

[Real] First coordinate of the 2D representation of a surface in UV space.

v

[Real] Second coordinate of the 2D representation of a surface in UV space.

Returns

UnitVector3D

Unit vector is normal to the surface at the given UV coordinates.

`ReversedTrimmedSurface.project_point`(*point*: *ansys.geometry.core.math.point.Point3D*) → *ansys.geometry.core.shapes_surfaces_surface_evaluation.SurfaceEvaluation*

Project a point onto the surface and evaluate it at that location.

Parameters

point

[Point3D] Point to project onto the surface.

Returns**SurfaceEvaluation**

Resulting evaluation.

Description

Provides the TrimmedSurface class.

Description

Provides the PyAnsys Geometry surface subpackage.

The box_uv.py module**Summary****Classes**

<i>BoxUV</i>	Provides the implementation for BoxUV class.
--------------	--

Enums

<i>LocationUV</i>	Provides the enumeration for indicating locations for BoxUV.
-------------------	--

BoxUV

```
class ansys.geometry.core.shapes.box_uv.BoxUV(range_u:
    ansys.geometry.core.shapes.parameterization.Interval =
    None, range_v:
    ansys.geometry.core.shapes.parameterization.Interval =
    None)
```

Provides the implementation for BoxUV class.

Overview**Methods**

<i>is_empty</i>	Check if this BoxUV is empty.
<i>proportion</i>	Evaluate the BoxUV at the given proportions.
<i>get_center</i>	Evaluate the this BoxUV in the center.
<i>is_negative</i>	Check whether the BoxUV is negative.
<i>contains</i>	Check whether the BoxUV contains a given u and v pair parameter.
<i>inflate</i>	Enlarge the BoxUV u and v intervals by deltas.
<i>get_corner</i>	Get the corner location of the BoxUV.

Properties

<i>interval_u</i>	u interval.
<i>interval_v</i>	v interval.

Special methods

<code>__eq__</code>	Check whether two BoxUV instances are equal.
<code>__ne__</code>	Check whether two BoxUV instances are not equal.

Import detail

```
from ansys.geometry.core.shapes.box_uv import BoxUV
```

Property detail

property `BoxUV.interval_u`: *ansys.geometry.core.shapes.parameterization.Interval*
u interval.

property `BoxUV.interval_v`: *ansys.geometry.core.shapes.parameterization.Interval*
v interval.

Method detail

`BoxUV.__eq__(other: object) → bool`
Check whether two BoxUV instances are equal.

`BoxUV.__ne__(other: object) → bool`
Check whether two BoxUV instances are not equal.

`BoxUV.is_empty()`
Check if this BoxUV is empty.

`BoxUV.proportion(prop_u: ansys.geometry.core.typing.Real, prop_v: ansys.geometry.core.typing.Real) → ansys.geometry.core.shapes.parameterization.ParamUV`
Evaluate the BoxUV at the given proportions.

`BoxUV.get_center()` → *ansys.geometry.core.shapes.parameterization.ParamUV*
Evaluate the this BoxUV in the center.

`BoxUV.is_negative(tolerance_u: ansys.geometry.core.typing.Real, tolerance_v: ansys.geometry.core.typing.Real) → bool`
Check whether the BoxUV is negative.

`BoxUV.contains(param: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`
Check whether the BoxUV contains a given u and v pair parameter.

`BoxUV.inflate(delta_u: ansys.geometry.core.typing.Real, delta_v: ansys.geometry.core.typing.Real) → BoxUV`
Enlarge the BoxUV u and v intervals by deltas.

`BoxUV.get_corner(location: LocationUV) → ansys.geometry.core.shapes.parameterization.ParamUV`
Get the corner location of the BoxUV.

LocationUV

class `ansys.geometry.core.shapes.box_uv.LocationUV(*args, **kws)`

Bases: `enum.Enum`

Provides the enumeration for indicating locations for BoxUV.

Overview

Attributes

<i>TopLeft</i>
<i>TopCenter</i>
<i>TopRight</i>
<i>BottomLeft</i>
<i>BottomCenter</i>
<i>BottomRight</i>
<i>LeftCenter</i>
<i>RightCenter</i>
<i>Center</i>

Import detail

```
from ansys.geometry.core.shapes.box_uv import LocationUV
```

Attribute detail

`LocationUV.TopLeft` = 1

`LocationUV.TopCenter` = 2

`LocationUV.TopRight` = 3

`LocationUV.BottomLeft` = 4

`LocationUV.BottomCenter` = 5

`LocationUV.BottomRight` = 6

`LocationUV.LeftCenter` = 7

`LocationUV.RightCenter` = 8

`LocationUV.Center` = 9

Description

Provides the BoxUV class.

The parameterization.py module

Summary

Classes

<i>ParamUV</i>	Parameter class containing 2 parameters: (u, v).
<i>Interval</i>	Interval class that defines a range of values.
<i>Parameterization</i>	Parameterization class describes the parameters of a specific geometry.

Enums

<i>ParamForm</i>	ParamForm enum class that defines the form of a Parameterization.
<i>ParamType</i>	ParamType enum class that defines the type of a Parameterization.

ParamUV

class `ansys.geometry.core.shapes.parameterization.ParamUV`(*u: ansys.geometry.core.typing.Real, v: ansys.geometry.core.typing.Real*)

Parameter class containing 2 parameters: (u, v).

Parameters

u	[Real] U parameter.
v	[Real] V parameter.

Notes

Likened to a 2D point in UV space. Used as an argument in parametric surface evaluations. This matches the service implementation for the Geometry service.

Overview

Properties

u	U parameter.
v	V parameter.

Special methods

<code>__add__</code>	Add the u and v components of the other ParamUV to this ParamUV.
<code>__sub__</code>	Subtract the u and v components of a ParamUV from this ParamUV.
<code>__mul__</code>	Multiplies the u and v components of this ParamUV by a ParamUV.
<code>__truediv__</code>	Divides the u and v components of this ParamUV by a ParamUV.
<code>__iter__</code>	Iterate a ParamUV.
<code>__repr__</code>	Represent the ParamUV as a string.

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamUV
```

Property detail

property `ParamUV.u`: `ansys.geometry.core.typing.Real`

U parameter.

property `ParamUV.v`: `ansys.geometry.core.typing.Real`

V parameter.

Method detail

`ParamUV.__add__(other: ParamUV) → ParamUV`

Add the u and v components of the other ParamUV to this ParamUV.

Parameters**other**

[ParamUV] The parameters to add these parameters.

Returns**ParamUV**

The sum of the parameters.

`ParamUV.__sub__(other: ParamUV) → ParamUV`

Subtract the u and v components of a ParamUV from this ParamUV.

Parameters**other**

[ParamUV] The parameters to subtract from these parameters.

Returns**ParamUV**

The difference of the parameters.

`ParamUV.__mul__(other: ParamUV) → ParamUV`

Multiplies the u and v components of this ParamUV by a ParamUV.

Parameters**other**

[ParamUV] The parameters to multiply by these parameters.

Returns**ParamUV**

The product of the parameters.

`ParamUV.__truediv__(other: ParamUV) → ParamUV`

Divides the u and v components of this ParamUV by a ParamUV.

Parameters**other**

[ParamUV] The parameters to divide these parameters by.

Returns**ParamUV**

The quotient of the parameters.

`ParamUV.__iter__()`

Iterate a ParamUV.

`ParamUV.__repr__()` → str

Represent the ParamUV as a string.

Interval

class `ansys.geometry.core.shapes.parameterization.Interval`(*start: ansys.geometry.core.typing.Real, end: ansys.geometry.core.typing.Real*)

Interval class that defines a range of values.

Parameters

- start**
[Real] Start value of the interval.
- end**
[Real] End value of the interval.

Overview

Methods

<code>is_open</code>	If the interval is open (-inf, inf).
<code>is_closed</code>	If the interval is closed. Neither value is inf or -inf.
<code>is_empty</code>	Check if the current interval is empty.
<code>get_span</code>	Get the quantity contained by the interval.
<code>get_relative_val</code>	Get an evaluation property of the interval, used in BoxUV.
<code>is_negative</code>	Indicate whether the current interval is negative.
<code>self_unite</code>	Get the union of two intervals and update the current interval.
<code>self_intersect</code>	Get the intersection of two intervals and update the current one.
<code>contains_value</code>	Check if the current interval contains the value <code>t</code> .
<code>contains</code>	Check if interval contains value <code>t</code> using default accuracy.
<code>inflate</code>	Enlarge the current interval by the given delta value.

Properties

<code>start</code>	Start value of the interval.
<code>end</code>	End value of the interval.

Attributes

<code>not_empty</code>

Static methods

<code>unite</code>	Get the union of two intervals.
<code>intersect</code>	Get the intersection of two intervals.

Special methods

<code>__eq__</code>	Compare two intervals.
<code>__repr__</code>	Represent the interval as a string.

Import detail

```
from ansys.geometry.core.shapes.parameterization import Interval
```

Property detail

property `Interval.start: ansys.geometry.core.typing.Real`

Start value of the interval.

property `Interval.end: ansys.geometry.core.typing.Real`

End value of the interval.

Attribute detail

`Interval.not_empty = True`

Method detail

`Interval.__eq__(other: object)`

Compare two intervals.

`Interval.is_open() → bool`

If the interval is open (-inf, inf).

Returns

bool

True if both ends of the interval are negative and positive infinity respectively.

`Interval.is_closed() → bool`

If the interval is closed. Neither value is inf or -inf.

Returns

bool

True if neither bound of the interval is infinite.

`Interval.is_empty() → bool`

Check if the current interval is empty.

Returns

bool

True when the interval is empty, False otherwise.

`Interval.get_span() → ansys.geometry.core.typing.Real`

Get the quantity contained by the interval.

The interval must be closed.

Returns

Real

The difference between the end and start of the interval.

`Interval.get_relative_val(t: ansys.geometry.core.typing.Real) → ansys.geometry.core.typing.Real`

Get an evaluation property of the interval, used in BoxUV.

Parameters

t
[Real] Offset to evaluate the interval at.

Returns

Real
Actual value according to the offset.

`Interval.is_negative(tolerance: ansys.geometry.core.typing.Real) → bool`

Indicate whether the current interval is negative.

Parameters

tolerance
[Real] Accepted range because the data type of the interval could be in doubles.

Returns

bool
True if the interval is negative, False otherwise.

`static Interval.unite(first: Interval, second: Interval) → Interval`

Get the union of two intervals.

Parameters

first
[Interval] First interval.

second
[Interval] Second interval.

Returns

Interval
Union of the two intervals.

`Interval.self_unite(other: Interval) → None`

Get the union of two intervals and update the current interval.

Parameters

other
[Interval] Interval to unite with.

`static Interval.intersect(first: Interval, second: Interval, tolerance: ansys.geometry.core.typing.Real) → Interval`

Get the intersection of two intervals.

Parameters

first
[Interval] First interval.

second
[Interval] Second interval.

Returns

Interval
Intersection of the two intervals.

`Interval.self_intersect(other: Interval, tolerance: ansys.geometry.core.typing.Real) → None`

Get the intersection of two intervals and update the current one.

Parameters

other

[Interval] Interval to intersect with.

tolerance

[Real] Accepted range of error given that the interval could be in float values.

`Interval.contains_value(t: ansys.geometry.core.typing.Real, accuracy: ansys.geometry.core.typing.Real) → bool`

Check if the current interval contains the value `t`.

Parameters

t

[Real] Value of interest.

accuracy

[Real] Accepted range of error given that the interval could be in float values.

Returns

bool

True if the interval contains the value, False otherwise.

`Interval.contains(t: ansys.geometry.core.typing.Real) → bool`

Check if interval contains value `t` using default accuracy.

Parameters

t

[Real] Value of interest.

Returns

bool

True if the interval contains the value, False otherwise.

`Interval.inflate(delta: ansys.geometry.core.typing.Real) → Interval`

Enlarge the current interval by the given delta value.

`Interval.__repr__()` → str

Represent the interval as a string.

Parameterization

`class ansys.geometry.core.shapes.parameterization.Parameterization(form: ParamForm, type: ParamType, interval: Interval)`

Parameterization class describes the parameters of a specific geometry.

Parameters

form

[ParamForm] Form of the parameterization.

type

[ParamType] Type of the parameterization.

interval

[Interval] Interval of the parameterization.

Overview**Properties**

<i>form</i>	The form of the parameterization.
<i>type</i>	The type of the parameterization.
<i>interval</i>	The interval of the parameterization.

Special methods

<code>__repr__</code>	Represent the Parameterization as a string.
-----------------------	---

Import detail

```
from ansys.geometry.core.shapes.parameterization import Parameterization
```

Property detail

property Parameterization.**form**: *ParamForm*

The form of the parameterization.

property Parameterization.**type**: *ParamType*

The type of the parameterization.

property Parameterization.**interval**: *Interval*

The interval of the parameterization.

Method detail

Parameterization.**__repr__**() → str

Represent the Parameterization as a string.

ParamForm

class ansys.geometry.core.shapes.parameterization.**ParamForm**(*args, **kwargs)

Bases: `enum.Enum`

ParamForm enum class that defines the form of a Parameterization.

Overview**Attributes**

<i>OPEN</i>
<i>CLOSED</i>
<i>PERIODIC</i>
<i>OTHER</i>

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamForm
```

Attribute detail

ParamForm.OPEN = 1

ParamForm.CLOSED = 2

ParamForm.PERIODIC = 3

ParamForm.OTHER = 4

ParamType

```
class ansys.geometry.core.shapes.parameterization.ParamType(*args, **kwargs)
```

Bases: `enum.Enum`

ParamType enum class that defines the type of a Parameterization.

Overview

Attributes

<i>LINEAR</i>
<i>CIRCULAR</i>
<i>OTHER</i>

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamType
```

Attribute detail

ParamType.LINEAR = 1

ParamType.CIRCULAR = 2

ParamType.OTHER = 3

Description

Provides the parametrization-related classes.

Description

Provides the PyAnsys Geometry geometry subpackage.

The sketch package

Summary

Submodules

<i>arc</i>	Provides for creating and managing an arc.
<i>box</i>	Provides for creating and managing a box (quadrilateral).
<i>circle</i>	Provides for creating and managing a circle.
<i>edge</i>	Provides for creating and managing an edge.
<i>ellipse</i>	Provides for creating and managing an ellipse.
<i>face</i>	Provides for creating and managing a face (closed 2D sketch).
<i>gears</i>	Module for creating and managing gears.
<i>polygon</i>	Provides for creating and managing a polygon.
<i>segment</i>	Provides for creating and managing a segment.
<i>sketch</i>	Provides for creating and managing a sketch.
<i>slot</i>	Provides for creating and managing a slot.
<i>trapezoid</i>	Provides for creating and managing a trapezoid.
<i>triangle</i>	Provides for creating and managing a triangle.

The arc.py module

Summary

Classes

<i>Arc</i>	Provides for modeling an arc.
------------	-------------------------------

Arc

```
class ansys.geometry.core.sketch.arc.Arc(start: ansys.geometry.core.math.point.Point2D, end:
    ansys.geometry.core.math.point.Point2D, center:
    ansys.geometry.core.math.point.Point2D, clockwise: bool =
    False)
```

Bases: *ansys.geometry.core.sketch.edge.SketchEdge*

Provides for modeling an arc.

Parameters

start

[Point2D] Starting point of the arc.

end

[Point2D] Ending point of the arc.

center

[Point2D] Center point of the arc.

clockwise

[bool, default: `False`] Whether the arc spans the clockwise angle between the start and end points. When `False` (default), the arc spans the counter-clockwise angle. When `True`, the arc spans the clockwise angle.

Overview

Constructors

<code>from_three_points</code>	Create an arc from three given points.
<code>from_start_end_and_radius</code>	Create an arc from a starting point, an ending point, and a radius.
<code>from_start_center_and_angle</code>	Create an arc from a starting point, a center point, and an angle.

Properties

<code>start</code>	Starting point of the arc line.
<code>end</code>	Ending point of the arc line.
<code>center</code>	Center point of the arc.
<code>length</code>	Length of the arc.
<code>radius</code>	Radius of the arc.
<code>angle</code>	Angle of the arc.
<code>is_clockwise</code>	Flag indicating whether the rotation of the angle is clockwise.
<code>sector_area</code>	Area of the sector of the arc.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Special methods

<code>__eq__</code>	Equals operator for the Arc class.
<code>__ne__</code>	Not equals operator for the Arc class.

Import detail

```
from ansys.geometry.core.sketch.arc import Arc
```

Property detail

property Arc.start: *ansys.geometry.core.math.point.Point2D*

Starting point of the arc line.

property Arc.end: *ansys.geometry.core.math.point.Point2D*

Ending point of the arc line.

property Arc.center: *ansys.geometry.core.math.point.Point2D*

Center point of the arc.

property Arc.length: *pint.Quantity*

Length of the arc.

property Arc.radius: *pint.Quantity*

Radius of the arc.

property Arc.angle: *pint.Quantity*

Angle of the arc.

property Arc.is_clockwise: *bool*

Flag indicating whether the rotation of the angle is clockwise.

Returns**bool**

True if the sense of rotation is clockwise. False if the sense of rotation is counter-clockwise.

property `Arc.sector_area`: **pint.Quantity**

Area of the sector of the arc.

property `Arc.visualization_polydata`: **pyvista.PolyData**

VTK polydata representation for PyVista visualization.

Returns**pyvista.PolyData**

VTK pyvista.Polydata configuration.

Notes

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Method detail`Arc.__eq__(other: Arc) → bool`Equals operator for the `Arc` class.`Arc.__ne__(other: Arc) → bool`Not equals operator for the `Arc` class.**classmethod** `Arc.from_three_points`(*start*: `ansys.geometry.core.math.point.Point2D`, *inter*: `ansys.geometry.core.math.point.Point2D`, *end*: `ansys.geometry.core.math.point.Point2D`)

Create an arc from three given points.

Parameters**start**[`Point2D`] Starting point of the arc.**inter**[`Point2D`] Intermediate point (location) of the arc.**end**[`Point2D`] Ending point of the arc.**Returns****Arc**

Arc generated from the three points.

classmethod `Arc.from_start_end_and_radius`(*start*: `ansys.geometry.core.math.point.Point2D`, *end*: `ansys.geometry.core.math.point.Point2D`, *radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *convex_arc*: `bool` = `False`, *clockwise*: `bool` = `False`)

Create an arc from a starting point, an ending point, and a radius.

Parameters**start**[`Point2D`] Starting point of the arc.

end

[Point2D] Ending point of the arc.

radius

[Quantity | Distance | Real] Radius of the arc.

convex_arc

[bool, default: `False`] Whether the arc is convex. The default is `False`. When `False`, the arc is concave. When `True`, the arc is convex.

clockwise

[bool, default: `False`] Whether the arc spans the clockwise angle between the start and end points. When `False`, the arc spans the counter-clockwise angle. When `True`, the arc spans the clockwise angle.

Returns**Arc**

Arc generated from the three points.

classmethod `Arc.from_start_center_and_angle`(*start*: ansys.geometry.core.math.point.Point2D, *center*: ansys.geometry.core.math.point.Point2D, *angle*: ansys.geometry.core.misc.measurements.Angle | *pint.Quantity* | *ansys.geometry.core.typing.Real*, *clockwise*: *bool = False*)

Create an arc from a starting point, a center point, and an angle.

Parameters**start**

[Point2D] Starting point of the arc.

center

[Point2D] Center point of the arc.

angle

[Angle | Quantity | Real] Angle of the arc.

clockwise

[bool, default: `False`] Whether the provided angle should be considered clockwise. When `False`, the angle is considered counter-clockwise. When `True`, the angle is considered clockwise.

Returns**Arc**

Arc generated from the three points.

Description

Provides for creating and managing an arc.

The `box.py` module**Summary****Classes**

Box	Provides for modeling a box.
------------	------------------------------

Box

```
class ansys.geometry.core.sketch.box.Box(center: ansys.geometry.core.math.point.Point2D, width:
    pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance |
    ansys.geometry.core.typing.Real, height: pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance |
    ansys.geometry.core.typing.Real, angle: pint.Quantity |
    ansys.geometry.core.misc.measurements.Angle |
    ansys.geometry.core.typing.Real = 0)
```

Bases: *ansys.geometry.core.sketch.face.SketchFace*

Provides for modeling a box.

Parameters

center: Point2D

Center point of the box.

width

[Quantity | Distance | Real] Width of the box.

height

[Quantity | Distance | Real] Height of the box.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Overview

Properties

<i>center</i>	Center point of the box.
<i>width</i>	Width of the box.
<i>height</i>	Height of the box.
<i>perimeter</i>	Perimeter of the box.
<i>area</i>	Area of the box.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.box import Box
```

Property detail

property Box.*center*: *ansys.geometry.core.math.point.Point2D*

Center point of the box.

property Box.*width*: *pint.Quantity*

Width of the box.

property Box.*height*: *pint.Quantity*

Height of the box.

property `Box.perimeter`: `pint.Quantity`

Perimeter of the box.

property `Box.area`: `pint.Quantity`

Area of the box.

property `Box.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Description

Provides for creating and managing a box (quadrilateral).

The `circle.py` module

Summary

Classes

<code>SketchCircle</code>	Provides for modeling a circle.
---------------------------	---------------------------------

SketchCircle

```
class ansys.geometry.core.sketch.circle.SketchCircle(center:
    ansys.geometry.core.math.point.Point2D,
    radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real, plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`, `ansys.geometry.core.shapes.curves.circle.Circle`

Provides for modeling a circle.

Parameters

center: `Point2D`

Center point of the circle.

radius

[`Quantity` | `Distance` | `Real`] Radius of the circle.

plane

[`Plane`, optional] Plane containing the sketched circle, which is the global XY plane by default.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing the SketchCircle objects.
---------------------------	---

Properties

<code>center</code>	Center of the circle.
<code>perimeter</code>	Perimeter of the circle.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.circle import SketchCircle
```

Property detail

property SketchCircle.`center`: `ansys.geometry.core.math.point.Point2D`

Center of the circle.

property SketchCircle.`perimeter`: `pint.Quantity`

Perimeter of the circle.

Notes

This property resolves the dilemma between using the `SkethFace.perimeter` property and the `Circle.perimeter` property.

property SketchCircle.`visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`
VTK pyvista.Polydata configuration.

Method detail

SketchCircle.`plane_change`(`plane`: `ansys.geometry.core.math.plane.Plane`) → `None`

Redefine the plane containing the SketchCircle objects.

Parameters

plane
[Plane] Desired new plane that is to contain the sketched circle.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing a circle.

The `edge.py` module

Summary

Classes

<code>SketchEdge</code>	Provides for modeling edges forming sketched shapes.
-------------------------	--

SketchEdge

class `ansys.geometry.core.sketch.edge.SketchEdge`

Provides for modeling edges forming sketched shapes.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing <code>SketchEdge</code> objects.
---------------------------	--

Properties

<code>start</code>	Starting point of the edge.
<code>end</code>	Ending point of the edge.
<code>length</code>	Length of the edge.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.edge import SketchEdge
```

Property detail

property `SketchEdge.start`: `ansys.geometry.core.math.point.Point2D`

Abstractmethod

Starting point of the edge.

property `SketchEdge.end`: `ansys.geometry.core.math.point.Point2D`

Abstractmethod

Ending point of the edge.

property `SketchEdge.length`: `pint.Quantity`

Abstractmethod

Length of the edge.

property SketchEdge.**visualization_polydata**: `pyvista.PolyData`

Abstractmethod

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Method detail

SketchEdge.**plane_change**(*plane*: `ansys.geometry.core.math.plane.Plane`) → `None`

Redefine the plane containing SketchEdge objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched edge.

Notes

This implies that their 3D definition might suffer changes. By default, this method does nothing. It is required to be implemented in child SketchEdge classes.

Description

Provides for creating and managing an edge.

The ellipse.py module

Summary

Classes

<code>SketchEllipse</code> Provides for modeling an ellipse.
--

SketchEllipse

```
class ansys.geometry.core.sketch.ellipse.SketchEllipse(center:
    ansys.geometry.core.math.point.Point2D,
    major_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real,
    minor_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real, angle:
    pint.Quantity | an-
    sys.geometry.core.misc.measurements.Angle
    | ansys.geometry.core.typing.Real = 0,
    plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`, `ansys.geometry.core.shapes.curves.ellipse.Ellipse`

Provides for modeling an ellipse.

Parameters

center: Point2D

Center point of the ellipse.

major_radius

[Quantity | Distance | Real] Major radius of the ellipse.

minor_radius

[Quantity | Distance | Real] Minor radius of the ellipse.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

plane

[Plane, optional] Plane containing the sketched ellipse, which is the global XY plane by default.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing SketchEllipse objects.
---------------------------	--

Properties

<code>center</code>	Center point of the ellipse.
<code>angle</code>	Orientation angle of the ellipse.
<code>perimeter</code>	Perimeter of the circle.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.ellipse import SketchEllipse
```

Property detail

property `SketchEllipse.center`: `ansys.geometry.core.math.point.Point2D`

Center point of the ellipse.

property `SketchEllipse.angle`: `pint.Quantity`

Orientation angle of the ellipse.

property `SketchEllipse.perimeter`: `pint.Quantity`

Perimeter of the circle.

Notes

This property resolves the dilemma between using the `SketchFace.perimeter` property and the `Ellipse.perimeter` property.

property `SketchEllipse.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the *X/Y* plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Method detail

`SketchEllipse.plane_change(plane: ansys.geometry.core.math.plane.Plane) → None`

Redefine the plane containing `SketchEllipse` objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched ellipse.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing an ellipse.

The `face.py` module

Summary

Classes

<code>SketchFace</code> Provides for modeling a face.

SketchFace

`class ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling a face.

Overview

Methods

<code>plane_change</code> Redefine the plane containing <code>SketchFace</code> objects.
--

Properties

<code>edges</code>	List of all component edges forming the face.
<code>perimeter</code>	Perimeter of the face.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.face import SketchFace
```

Property detail

property `SketchFace.edges`: `list[ansys.geometry.core.sketch.edge.SketchEdge]`

List of all component edges forming the face.

property `SketchFace.perimeter`: `pint.Quantity`

Perimeter of the face.

property `SketchFace.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Method detail

`SketchFace.plane_change`(*plane*: `ansys.geometry.core.math.plane.Plane`) → `None`

Redefine the plane containing `SketchFace` objects.

Parameters

plane

[`Plane`] Desired new plane that is to contain the sketched face.

Notes

This implies that their 3D definition might suffer changes. This method does nothing by default. It is required to be implemented in child `SketchFace` classes.

Description

Provides for creating and managing a face (closed 2D sketch).

The `gears.py` module

Summary

Classes

<code>Gear</code>	Provides the base class for sketching gears.
<code>DummyGear</code>	Provides the dummy class for sketching gears.
<code>SpurGear</code>	Provides the class for sketching spur gears.

Gear

class `ansys.geometry.core.sketch.gears.Gear`

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides the base class for sketching gears.

Overview

Properties

<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.
-------------------------------------	--

Import detail

```
from ansys.geometry.core.sketch.gears import Gear
```

Property detail

property `Gear.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`
VTK `pyvista.Polydata` configuration.

DummyGear

class `ansys.geometry.core.sketch.gears.DummyGear` (*origin*: `ansys.geometry.core.math.point.Point2D`, *outer_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *inner_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *n_teeth*: `int`)

Bases: `Gear`

Provides the dummy class for sketching gears.

Parameters

origin
[`Point2D`] Origin of the gear.

outer_radius
[`Quantity` | `Distance` | `Real`] Outer radius of the gear.

inner_radius

[Quantity | Distance | Real] Inner radius of the gear.

n_teeth

[int] Number of teeth of the gear.

Import detail

```
from ansys.geometry.core.sketch.gears import DummyGear
```

SpurGear

```
class ansys.geometry.core.sketch.gears.SpurGear(origin: ansys.geometry.core.math.point.Point2D,
module: ansys.geometry.core.typing.Real,
pressure_angle: pint.Quantity |
ansys.geometry.core.misc.measurements.Angle |
ansys.geometry.core.typing.Real, n_teeth: int)
```

Bases: Gear

Provides the class for sketching spur gears.

Parameters**origin**

[Point2D] Origin of the spur gear.

module

[Real] Module of the spur gear. This is also the ratio between the pitch circle diameter in millimeters and the number of teeth.

pressure_angle

[Quantity | Angle | Real] Pressure angle of the spur gear.

n_teeth

[int] Number of teeth of the spur gear.

Overview**Properties**

<i>origin</i>	Origin of the spur gear.
<i>module</i>	Module of the spur gear.
<i>pressure_angle</i>	Pressure angle of the spur gear.
<i>n_teeth</i>	Number of teeth of the spur gear.
<i>ref_diameter</i>	Reference diameter of the spur gear.
<i>base_diameter</i>	Base diameter of the spur gear.
<i>addendum</i>	Addendum of the spur gear.
<i>dedendum</i>	Dedendum of the spur gear.
<i>tip_diameter</i>	Tip diameter of the spur gear.
<i>root_diameter</i>	Root diameter of the spur gear.

Import detail

```
from ansys.geometry.core.sketch.gears import SpurGear
```

Property detail

property `SpurGear.origin`: `ansys.geometry.core.math.point.Point2D`

Origin of the spur gear.

property `SpurGear.module`: `ansys.geometry.core.typing.Real`

Module of the spur gear.

property `SpurGear.pressure_angle`: `pint.Quantity`

Pressure angle of the spur gear.

property `SpurGear.n_teeth`: `int`

Number of teeth of the spur gear.

property `SpurGear.ref_diameter`: `ansys.geometry.core.typing.Real`

Reference diameter of the spur gear.

property `SpurGear.base_diameter`: `ansys.geometry.core.typing.Real`

Base diameter of the spur gear.

property `SpurGear.addendum`: `ansys.geometry.core.typing.Real`

Addendum of the spur gear.

property `SpurGear.dedendum`: `ansys.geometry.core.typing.Real`

Dedendum of the spur gear.

property `SpurGear.tip_diameter`: `ansys.geometry.core.typing.Real`

Tip diameter of the spur gear.

property `SpurGear.root_diameter`: `ansys.geometry.core.typing.Real`

Root diameter of the spur gear.

Description

Module for creating and managing gears.

The `polygon.py` module

Summary

Classes

<i>Polygon</i>	Provides for modeling regular polygons.
----------------	---

Polygon

```
class ansys.geometry.core.sketch.polygon.Polygon(center: ansys.geometry.core.math.point.Point2D,
                                                inner_radius: pint.Quantity |
                                                ansys.geometry.core.misc.measurements.Distance |
                                                ansys.geometry.core.typing.Real, sides: int, angle:
                                                pint.Quantity |
                                                ansys.geometry.core.misc.measurements.Angle |
                                                ansys.geometry.core.typing.Real = 0)
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling regular polygons.

Parameters

center: Point2D

Center point of the circle.

inner_radius

[Quantity | Distance | Real] Inner radius (apothem) of the polygon.

sides

[int] Number of sides of the polygon.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Overview

Properties

<code>center</code>	Center point of the polygon.
<code>inner_radius</code>	Inner radius (apothem) of the polygon.
<code>n_sides</code>	Number of sides of the polygon.
<code>angle</code>	Orientation angle of the polygon.
<code>length</code>	Side length of the polygon.
<code>outer_radius</code>	Outer radius of the polygon.
<code>perimeter</code>	Perimeter of the polygon.
<code>area</code>	Area of the polygon.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.polygon import Polygon
```

Property detail

property Polygon.`center`: `ansys.geometry.core.math.point.Point2D`

Center point of the polygon.

property Polygon.`inner_radius`: `pint.Quantity`

Inner radius (apothem) of the polygon.

property Polygon.`n_sides`: `int`

Number of sides of the polygon.

property Polygon.**angle**: `pint.Quantity`

Orientation angle of the polygon.

property Polygon.**length**: `pint.Quantity`

Side length of the polygon.

property Polygon.**outer_radius**: `pint.Quantity`

Outer radius of the polygon.

property Polygon.**perimeter**: `pint.Quantity`

Perimeter of the polygon.

property Polygon.**area**: `pint.Quantity`

Area of the polygon.

property Polygon.**visualization_polydata**: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Description

Provides for creating and managing a polygon.

The `segment.py` module

Summary

Classes

<code>SketchSegment</code>	Provides segment representation of a line.
----------------------------	--

SketchSegment

```
class ansys.geometry.core.sketch.segment.SketchSegment(start:
    ansys.geometry.core.math.point.Point2D,
    end:
    ansys.geometry.core.math.point.Point2D,
    plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```

Bases: `ansys.geometry.core.sketch.edge.SketchEdge`, `ansys.geometry.core.shapes.curves.Line.Line`

Provides segment representation of a line.

Parameters

start
[Point2D] Starting point of the line segment.

end
[Point2D] Ending point of the line segment.

plane

[Plane, optional] Plane containing the sketched circle, which is the global XY plane by default.

Overview**Methods**

<code>plane_change</code>	Redefine the plane containing SketchSegment objects.
---------------------------	--

Properties

<code>start</code>	Starting point of the segment.
<code>end</code>	Ending point of the segment.
<code>length</code>	Length of the segment.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Special methods

<code>__eq__</code>	Equals operator for the SketchSegment class.
<code>__ne__</code>	Not equals operator for the SketchSegment class.

Import detail

```
from ansys.geometry.core.sketch.segment import SketchSegment
```

Property detail

property SketchSegment.start: `ansys.geometry.core.math.point.Point2D`
Starting point of the segment.

property SketchSegment.end: `ansys.geometry.core.math.point.Point2D`
Ending point of the segment.

property SketchSegment.length: `pint.Quantity`
Length of the segment.

property SketchSegment.visualization_polydata: `pyvista.PolyData`
VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`
VTK pyvista.Polydata configuration.

Method detail

SketchSegment.__eq__(other: SketchSegment) → bool
Equals operator for the SketchSegment class.

SketchSegment.__ne__(other: SketchSegment) → bool

Not equals operator for the SketchSegment class.

SketchSegment.plane_change(plane: ansys.geometry.core.math.plane.Plane) → None

Redefine the plane containing SketchSegment objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched segment.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing a segment.

The sketch.py module

Summary

Classes

<i>Sketch</i>	Provides for building 2D sketch elements.
---------------	---

Attributes

<i>SketchObject</i>	Type to refer to both SketchEdge and SketchFace.
---------------------	--

Sketch

class ansys.geometry.core.sketch.sketch.**Sketch**(plane: ansys.geometry.core.math.plane.Plane = Plane())

Provides for building 2D sketch elements.

Overview

Methods

<i>translate_sketch_plane</i>	Translate the origin location of the active sketch plane.
<i>translate_sketch_plane_by_offset</i>	Translate the origin location of the active sketch plane by offsets.
<i>translate_sketch_plane_by_distance</i>	Translate the origin location active sketch plane by distance.
<i>get</i>	Get a list of shapes with a given tag.
<i>face</i>	Add a sketch face to the sketch.
<i>edge</i>	Add a sketch edge to the sketch.
<i>select</i>	Add all objects that match provided tags to the current context.
<i>segment</i>	Add a segment sketch object to the sketch plane.
<i>segment_to_point</i>	Add a segment to the sketch plane starting from the previous end point.
<i>segment_from_point_and_vector</i>	Add a segment to the sketch starting from a given starting point.

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<code>segment_from_vector</code>	Add a segment to the sketch starting from the previous end point.
<code>arc</code>	Add an arc to the sketch plane.
<code>arc_to_point</code>	Add an arc to the sketch starting from the previous end point.
<code>arc_from_three_points</code>	Add an arc to the sketch plane from three given points.
<code>arc_from_start_end_and_radius</code>	Add an arc from the start, end points and a radius.
<code>arc_from_start_center_and_angle</code>	Add an arc from the start, center point, and angle.
<code>triangle</code>	Add a triangle to the sketch using given vertex points.
<code>trapezoid</code>	Add a trapezoid to the sketch using given vertex points.
<code>circle</code>	Add a circle to the plane at a given center.
<code>box</code>	Create a box on the sketch.
<code>slot</code>	Create a slot on the sketch.
<code>ellipse</code>	Create an ellipse on the sketch.
<code>polygon</code>	Create a polygon on the sketch.
<code>dummy_gear</code>	Create a dummy gear on the sketch.
<code>spur_gear</code>	Create a spur gear on the sketch.
<code>tag</code>	Add a tag to the active selection of sketch objects.
<code>plot</code>	Plot all objects of the sketch to the scene.
<code>plot_selection</code>	Plot the current selection to the scene.
<code>sketch_polydata</code>	Get polydata configuration for all objects of the sketch.
<code>sketch_polydata_faces</code>	Get polydata configuration for all faces of the sketch to the scene.
<code>sketch_polydata_edges</code>	Get polydata configuration for all edges of the sketch to the scene.

Properties

<code>plane</code>	Sketch plane configuration.
<code>edges</code>	List of all independently sketched edges.
<code>faces</code>	List of all independently sketched faces.

Import detail

```
from ansys.geometry.core.sketch.sketch import Sketch
```

Property detail

property `Sketch.plane`: `ansys.geometry.core.math.plane.Plane`

Sketch plane configuration.

property `Sketch.edges`: `list[ansys.geometry.core.sketch.edge.SketchEdge]`

List of all independently sketched edges.

Notes

Independently sketched edges are not assigned to a face. Face edges are not included in this list.

property `Sketch.faces`: `list[ansys.geometry.core.sketch.face.SketchFace]`

List of all independently sketched faces.

Method detail

`Sketch.translate_sketch_plane`(*translation*: `ansys.geometry.core.math.vector.Vector3D`) → *Sketch*

Translate the origin location of the active sketch plane.

Parameters**translation**

[`Vector3D`] Vector defining the translation. Meters is the expected unit.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.translate_sketch_plane_by_offset`(*x*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance = Quantity(0, DEFAULT_UNITS.LENGTH)`, *y*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance = Quantity(0, DEFAULT_UNITS.LENGTH)`, *z*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance = Quantity(0, DEFAULT_UNITS.LENGTH)`) → *Sketch*

Translate the origin location of the active sketch plane by offsets.

Parameters**x**

[`Quantity` | `Distance`, default: `Quantity(0, DEFAULT_UNITS.LENGTH)`] Amount to translate the origin of the x-direction.

y

[`Quantity` | `Distance`, default: `Quantity(0, DEFAULT_UNITS.LENGTH)`] Amount to translate the origin of the y-direction.

z

[`Quantity` | `Distance`, default: `Quantity(0, DEFAULT_UNITS.LENGTH)`] Amount to translate the origin of the z-direction.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.translate_sketch_plane_by_distance`(*direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *distance*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance`) → *Sketch*

Translate the origin location active sketch plane by distance.

Parameters**direction**

[`UnitVector3D`] Direction to translate the origin.

distance

[`Quantity` | `Distance`] Distance to translate the origin.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.get(tag: str) → list[SketchObject]`

Get a list of shapes with a given tag.

Parameters

tag

[str] Tag to query against.

`Sketch.face(face: ansys.geometry.core.sketch.face.SketchFace, tag: str | None = None) → Sketch`

Add a sketch face to the sketch.

Parameters

face

[SketchFace] Face to add.

tag

[str, default: None] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.edge(edge: ansys.geometry.core.sketch.edge.SketchEdge, tag: str | None = None) → Sketch`

Add a sketch edge to the sketch.

Parameters

edge

[SketchEdge] Edge to add.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.select(*tags: str) → Sketch`

Add all objects that match provided tags to the current context.

`Sketch.segment(start: ansys.geometry.core.math.point.Point2D, end: ansys.geometry.core.math.point.Point2D, tag: str | None = None) → Sketch`

Add a segment sketch object to the sketch plane.

Parameters

start

[Point2D] Starting point of the line segment.

end

[Point2D] Ending point of the line segment.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.segment_to_point`(*end*: ansys.geometry.core.math.point.Point2D, *tag*: *str* | *None* = *None*) → *Sketch*

Add a segment to the sketch plane starting from the previous end point.

Parameters

end

[Point2D] Ending point of the line segment.

tag

[*str*, default: *None*] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Notes

The starting point of the created edge is based upon the current context of the sketch, such as the end point of a previously added edge.

`Sketch.segment_from_point_and_vector`(*start*: ansys.geometry.core.math.point.Point2D, *vector*: ansys.geometry.core.math.vector.Vector2D, *tag*: *str* | *None* = *None*)

Add a segment to the sketch starting from a given starting point.

Parameters

start

[Point2D] Starting point of the line segment.

vector

[Vector2D] Vector defining the line segment. Vector magnitude determines the segment endpoint. Vector magnitude is assumed to be in the same unit as the starting point.

tag

[*str*, default: *None*] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Notes

Vector magnitude determines the segment endpoint. Vector magnitude is assumed to use the same unit as the starting point.

`Sketch.segment_from_vector`(*vector*: ansys.geometry.core.math.vector.Vector2D, *tag*: *str* | *None* = *None*)

Add a segment to the sketch starting from the previous end point.

Parameters

vector

[Vector2D] Vector defining the line segment.

tag

[*str*, default: *None*] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Notes

The starting point of the created edge is based upon the current context of the sketch, such as the end point of a previously added edge.

Vector magnitude determines the segment endpoint. Vector magnitude is assumed to use the same unit as the starting point in the previous context.

`Sketch.arc(start: ansys.geometry.core.math.point.Point2D, end: ansys.geometry.core.math.point.Point2D, center: ansys.geometry.core.math.point.Point2D, clockwise: bool = False, tag: str | None = None) → Sketch`

Add an arc to the sketch plane.

Parameters

start

[Point2D] Starting point of the arc.

end

[Point2D] Ending point of the arc.

center

[Point2D] Center point of the arc.

clockwise

[bool, default: False] Whether the arc spans the angle clockwise between the start and end points. When False (default), the arc spans the angle counter-clockwise. When True, the arc spans the angle clockwise.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.arc_to_point(end: ansys.geometry.core.math.point.Point2D, center: ansys.geometry.core.math.point.Point2D, clockwise: bool = False, tag: str | None = None) → Sketch`

Add an arc to the sketch starting from the previous end point.

Parameters

end

[Point2D] Ending point of the arc.

center

[Point2D] Center point of the arc.

clockwise

[bool, default: False] Whether the arc spans the angle clockwise between the start and end points. When False (default), the arc spans the angle counter-clockwise. When True, the arc spans the angle clockwise.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Notes

The starting point of the created edge is based upon the current context of the sketch, such as the end point of a previously added edge.

`Sketch.arc_from_three_points`(*start*: `ansys.geometry.core.math.point.Point2D`, *inter*: `ansys.geometry.core.math.point.Point2D`, *end*: `ansys.geometry.core.math.point.Point2D`, *tag*: `str` | `None = None`) → `Sketch`

Add an arc to the sketch plane from three given points.

Parameters

start

[`Point2D`] Starting point of the arc.

inter

[`Point2D`] Intermediate point (location) of the arc.

end

[`Point2D`] End point of the arc.

tag

[`str`, default: `None`] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.arc_from_start_end_and_radius`(*start*: `ansys.geometry.core.math.point.Point2D`, *end*: `ansys.geometry.core.math.point.Point2D`, *radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *convex_arc*: `bool = False`, *clockwise*: `bool = False`, *tag*: `str` | `None = None`) → `Sketch`

Add an arc from the start, end points and a radius.

Parameters

start

[`Point2D`] Starting point of the arc.

end

[`Point2D`] Ending point of the arc.

radius

[`Quantity` | `Distance` | `Real`] Radius of the arc.

convex_arc

[`bool`, default: `False`] Whether the arc is convex. The default is `False`. When `False`, the arc spans the concave version of the arc. When `True`, the arc spans the convex version of the arc.

clockwise

[`bool`, default: `False`] Whether the arc spans the angle clockwise between the start and end points. When `False`, the arc spans the angle counter-clockwise. When `True`, the arc spans the angle clockwise.

tag

[`str`, default: `None`] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.arc_from_start_center_and_angle` (*start*: `ansys.geometry.core.math.point.Point2D`, *center*: `ansys.geometry.core.math.point.Point2D`, *angle*: `pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real`, *clockwise*: `bool = False`, *tag*: `str | None = None`) → *Sketch*

Add an arc from the start, center point, and angle.

Parameters**start**

[`Point2D`] Starting point of the arc.

center

[`Point2D`] Center point of the arc.

angle

[`Quantity | Angle | Real`] Angle of the arc.

clockwise

[`bool`, default: `False`] Whether the arc spans the angle clockwise. The default is `False`. When `False`, the arc spans the angle counter-clockwise. When `True`, the arc spans the angle clockwise.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.triangle` (*point1*: `ansys.geometry.core.math.point.Point2D`, *point2*: `ansys.geometry.core.math.point.Point2D`, *point3*: `ansys.geometry.core.math.point.Point2D`, *tag*: `str | None = None`) → *Sketch*

Add a triangle to the sketch using given vertex points.

Parameters**point1**

[`Point2D`] Point that represents a vertex of the triangle.

point2

[`Point2D`] Point that represents a vertex of the triangle.

point3

[`Point2D`] Point that represents a vertex of the triangle.

tag

[`str`, default: `None`] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.trapezoid`(*base_width*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*, *height*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*, *base_angle*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Angle* | *ansys.geometry.core.typing.Real*, *base_asymmetric_angle*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Angle* | *ansys.geometry.core.typing.Real* | *None* = *None*, *center*: *ansys.geometry.core.math.point.Point2D* = *ZERO_POINT2D*, *angle*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Angle* | *ansys.geometry.core.typing.Real* = 0, *tag*: *str* | *None* = *None*) → *Sketch*

Add a trapezoid to the sketch using given vertex points.

Parameters

base_width

[*Quantity* | *Distance* | *Real*] Width of the lower base of the trapezoid.

height

[*Quantity* | *Distance* | *Real*] Height of the slot.

base_angle

[*Quantity* | *Distance* | *Real*] Angle for trapezoid generation. Represents the angle on the base of the trapezoid.

base_asymmetric_angle

[*Quantity* | *Angle* | *Real* | *None*, default: *None*] Asymmetrical angles on each side of the trapezoid. The default is *None*, in which case the trapezoid is symmetrical. If provided, the trapezoid is asymmetrical and the right corner angle at the base of the trapezoid is set to the provided value.

center: Point2D, default: ZERO_POINT2D

Center point of the trapezoid.

angle

[*Quantity* | *Angle* | *Real*, default: 0] Placement angle for orientation alignment.

tag

[*str*, default: *None*] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Notes

If an asymmetric base angle is defined, the base angle is applied to the left-most angle, and the asymmetric base angle is applied to the right-most angle.

`Sketch.circle`(*center*: *ansys.geometry.core.math.point.Point2D*, *radius*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*, *tag*: *str* | *None* = *None*) → *Sketch*

Add a circle to the plane at a given center.

Parameters

center: Point2D

Center point of the circle.

radius

[*Quantity* | *Distance* | *Real*] Radius of the circle.

tag

[*str*, default: *None*] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.box`(*center*: `ansys.geometry.core.math.point.Point2D`, *width*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *height*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *angle*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Angle` | `ansys.geometry.core.typing.Real = 0`, *tag*: `str` | `None = None`) → *Sketch*

Create a box on the sketch.

Parameters**center: Point2D**

Center point of the box.

width

[`Quantity` | `Distance` | `Real`] Width of the box.

height

[`Quantity` | `Distance` | `Real`] Height of the box.

angle

[`Quantity` | `Angle` | `Real`, default: 0] Placement angle for orientation alignment.

tag

[*str*, default: *None*] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.slot`(*center*: `ansys.geometry.core.math.point.Point2D`, *width*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *height*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *angle*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Angle` | `ansys.geometry.core.typing.Real = 0`, *tag*: `str` | `None = None`) → *Sketch*

Create a slot on the sketch.

Parameters**center: Point2D**

Center point of the slot.

width

[`Quantity` | `Distance` | `Real`] Width of the slot.

height

[`Quantity` | `Distance` | `Real`] Height of the slot.

angle

[`Quantity` | `Angle` | `Real`, default: 0] Placement angle for orientation alignment.

tag

[*str*, default: *None*] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.ellipse`(*center*: `ansys.geometry.core.math.point.Point2D`, *major_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *minor_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *angle*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Angle` | `ansys.geometry.core.typing.Real = 0`, *tag*: `str` | `None = None`) → *Sketch*

Create an ellipse on the sketch.

Parameters

center: Point2D

Center point of the ellipse.

major_radius

[`Quantity` | `Distance` | `Real`] Semi-major axis of the ellipse.

minor_radius

[`Quantity` | `Distance` | `Real`] Semi-minor axis of the ellipse.

angle

[`Quantity` | `Angle` | `Real`, default: 0] Placement angle for orientation alignment.

tag

[`str`, default: `None`] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.polygon`(*center*: `ansys.geometry.core.math.point.Point2D`, *inner_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *sides*: `int`, *angle*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Angle` | `ansys.geometry.core.typing.Real = 0`, *tag*: `str` | `None = None`) → *Sketch*

Create a polygon on the sketch.

Parameters

center: Point2D

Center point of the polygon.

inner_radius

[`Quantity` | `Distance` | `Real`] Inner radius (apothem) of the polygon.

sides

[`int`] Number of sides of the polygon.

angle

[`Quantity` | `Angle` | `Real`, default: 0] Placement angle for orientation alignment.

tag

[`str`, default: `None`] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.dummy_gear` (*origin*: `ansys.geometry.core.math.point.Point2D`, *outer_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *inner_radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *n_teeth*: `int`, *tag*: `str` | `None = None`) → `Sketch`

Create a dummy gear on the sketch.

Parameters

origin

[`Point2D`] Origin of the gear.

outer_radius

[`Quantity` | `Distance` | `Real`] Outer radius of the gear.

inner_radius

[`Quantity` | `Distance` | `Real`] Inner radius of the gear.

n_teeth

[`int`] Number of teeth of the gear.

tag

[`str`, default: `None`] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.spur_gear` (*origin*: `ansys.geometry.core.math.point.Point2D`, *module*: `ansys.geometry.core.typing.Real`, *pressure_angle*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Angle` | `ansys.geometry.core.typing.Real`, *n_teeth*: `int`, *tag*: `str` | `None = None`) → `Sketch`

Create a spur gear on the sketch.

Parameters

origin

[`Point2D`] Origin of the spur gear.

module

[`Real`] Module of the spur gear. This is also the ratio between the pitch circle diameter in millimeters and the number of teeth.

pressure_angle

[`Quantity` | `Angle` | `Real`] Pressure angle of the spur gear.

n_teeth

[`int`] Number of teeth of the spur gear.

tag

[`str`, default: `None`] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.tag` (*tag*: `str`) → `None`

Add a tag to the active selection of sketch objects.

Parameters

tag

[`str`] Tag to assign to the sketch objects.

`Sketch.plot`(*view_2d*: *bool* = *False*, *screenshot*: *str* | *None* = *None*, *use_frame*: *bool* | *None* = *None*,
selected_pd_objects: *list*[*pyvista.PolyData*] = *None*, ***plotting_options*: *dict* | *None*)

Plot all objects of the sketch to the scene.

Parameters

`view_2d`

[*bool*, default: *False*] Whether to represent the plot in a 2D format.

`screenshot`

[*str*, optional] Path for saving a screenshot of the image that is being represented.

`use_frame`

[*bool*, default: *None*] Whether to enables the use of `frame`. The default is *None*, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.

***plotting_options*

[*dict*, optional] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`Sketch.plot_selection`(*view_2d*: *bool* = *False*, *screenshot*: *str* | *None* = *None*, *use_frame*: *bool* | *None* = *None*,
***plotting_options*: *dict* | *None*)

Plot the current selection to the scene.

Parameters

`view_2d`

[*bool*, default: *False*] Whether to represent the plot in a 2D format.

`screenshot`

[*str*, optional] Path for saving a screenshot of the image that is being represented.

`use_frame`

[*bool*, default: *None*] Whether to enables the use of `frame`. The default is *None*, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.

***plotting_options*

[*dict*, optional] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`Sketch.sketch_polydata`() → *list*[*pyvista.PolyData*]

Get polydata configuration for all objects of the sketch.

Returns

list[*PolyData*]

List of the polydata configuration for all edges and faces in the sketch.

`Sketch.sketch_polydata_faces`() → *list*[*pyvista.PolyData*]

Get polydata configuration for all faces of the sketch to the scene.

Returns

list[*PolyData*]

List of the polydata configuration for faces in the sketch.

`Sketch.sketch_polydata_edges`() → *list*[*pyvista.PolyData*]

Get polydata configuration for all edges of the sketch to the scene.

Returns

list[*PolyData*]

List of the polydata configuration for edges in the sketch.

Description

Provides for creating and managing a sketch.

Module detail

sketch.SketchObject

Type to refer to both SketchEdge and SketchFace.

The slot.py module

Summary

Classes

<i>Slot</i>	Provides for modeling a 2D slot.
-------------	----------------------------------

Slot

```
class ansys.geometry.core.sketch.slot.Slot(center: ansys.geometry.core.math.point.Point2D, width:  
    pint.Quantity |  
    ansys.geometry.core.misc.measurements.Distance |  
    ansys.geometry.core.typing.Real, height: pint.Quantity |  
    ansys.geometry.core.misc.measurements.Distance |  
    ansys.geometry.core.typing.Real, angle: pint.Quantity |  
    ansys.geometry.core.misc.measurements.Angle |  
    ansys.geometry.core.typing.Real = 0)
```

Bases: *ansys.geometry.core.sketch.face.SketchFace*

Provides for modeling a 2D slot.

Parameters

center: :class:`Point2D <ansys.geometry.core.math.point.Point2D>`
Center point of the slot.

width
[Quantity | Distance | Real] Width of the slot main body.

height
[Quantity | Distance | Real] Height of the slot.

angle
[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Overview

Properties

<i>center</i>	Center of the slot.
<i>width</i>	Width of the slot.
<i>height</i>	Height of the slot.
<i>perimeter</i>	Perimeter of the slot.
<i>area</i>	Area of the slot.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.slot import Slot
```

Property detail

property Slot.**center**: *ansys.geometry.core.math.point.Point2D*

Center of the slot.

property Slot.**width**: *pint.Quantity*

Width of the slot.

property Slot.**height**: *pint.Quantity*

Height of the slot.

property Slot.**perimeter**: *pint.Quantity*

Perimeter of the slot.

property Slot.**area**: *pint.Quantity*

Area of the slot.

property Slot.**visualization_polydata**: *pyvista.PolyData*

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData

VTK *pyvista.Polydata* configuration.

Description

Provides for creating and managing a slot.

The `trapezoid.py` module

Summary

Classes

<i>Trapezoid</i>	Provides for modeling a 2D trapezoid.
------------------	---------------------------------------

Trapezoid

```

class ansys.geometry.core.sketch.trapezoid.Trapezoid(base_width: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real, height:
pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real, base_angle:
pint.Quantity |
ansys.geometry.core.misc.measurements.Angle
| ansys.geometry.core.typing.Real,
base_asymmetric_angle: pint.Quantity |
ansys.geometry.core.misc.measurements.Angle
| ansys.geometry.core.typing.Real | None =
None, center:
ansys.geometry.core.math.point.Point2D =
ZERO_POINT2D, angle: pint.Quantity |
ansys.geometry.core.misc.measurements.Angle
| ansys.geometry.core.typing.Real = 0)

```

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling a 2D trapezoid.

Parameters

base_width

[Quantity | Distance | Real] Width of the lower base of the trapezoid.

height

[Quantity | Distance | Real] Height of the slot.

base_angle

[Quantity | Distance | Real] Angle for trapezoid generation. Represents the angle on the base of the trapezoid.

base_asymmetric_angle

[Quantity | Angle | Real | None, default: None] Asymmetrical angles on each side of the trapezoid. The default is None, in which case the trapezoid is symmetrical. If provided, the trapezoid is asymmetrical and the right corner angle at the base of the trapezoid is set to the provided value.

center: Point2D, default: ZERO_POINT2D

Center point of the trapezoid.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Notes

If an asymmetric base angle is defined, the base angle is applied to the left-most angle, and the asymmetric base angle is applied to the right-most angle.

Overview

Properties

<code>center</code>	Center of the trapezoid.
<code>base_width</code>	Width of the trapezoid.
<code>height</code>	Height of the trapezoid.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.trapezoid import Trapezoid
```

Property detail

property Trapezoid.`center`: `ansys.geometry.core.math.point.Point2D`

Center of the trapezoid.

property Trapezoid.`base_width`: `pint.Quantity`

Width of the trapezoid.

property Trapezoid.`height`: `pint.Quantity`

Height of the trapezoid.

property Trapezoid.`visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a trapezoid.

The triangle.py module

Summary

Classes

<code>Triangle</code>	Provides for modeling 2D triangles.
-----------------------	-------------------------------------

Triangle

```
class ansys.geometry.core.sketch.triangle.Triangle(point1: ansys.geometry.core.math.point.Point2D,  
                                                    point2: ansys.geometry.core.math.point.Point2D,  
                                                    point3: ansys.geometry.core.math.point.Point2D)
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling 2D triangles.

Parameters

point1: Point2D

Point that represents a triangle vertex.

point2: Point2D

Point that represents a triangle vertex.

point3: Point2D

Point that represents a triangle vertex.

Overview**Properties**

<i>point1</i>	Triangle vertex 1.
<i>point2</i>	Triangle vertex 2.
<i>point3</i>	Triangle vertex 3.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.triangle import Triangle
```

Property detail

property Triangle.**point1**: *ansys.geometry.core.math.point.Point2D*

Triangle vertex 1.

property Triangle.**point2**: *ansys.geometry.core.math.point.Point2D*

Triangle vertex 2.

property Triangle.**point3**: *ansys.geometry.core.math.point.Point2D*

Triangle vertex 3.

property Triangle.**visualization_polydata**: *pyvista.PolyData*

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a triangle.

Description

PyAnsys Geometry sketch subpackage.

The tools package

Summary

Submodules

<code>check_geometry</code>	Module for repair tool message.
<code>measurement_tools</code>	Provides tools for measurement.
<code>prepare_tools</code>	Provides tools for preparing geometry for use with simulation.
<code>problem_areas</code>	The problem area definition.
<code>repair_tool_message</code>	Module for repair tool message.
<code>repair_tools</code>	Provides tools for repairing bodies.
<code>unsupported</code>	Unsupported functions for the PyAnsys Geometry library.

The `check_geometry.py` module

Summary

Classes

<code>GeometryIssue</code>	Provides return message for the repair tool methods.
<code>InspectResult</code>	Provides the result of the inspect geometry operation.

GeometryIssue

```
class ansys.geometry.core.tools.check_geometry.GeometryIssue(message_type: str, message_id: str,  
message: str, edges: list[str], faces:  
list[str])
```

Provides return message for the repair tool methods.

Overview

Properties

<code>message_type</code>	The type of the message (warning, error, info).
<code>message_id</code>	The identifier for the message.
<code>message</code>	The content of the message.
<code>edges</code>	The List of edges (if any) that are part of the issue.
<code>faces</code>	The List of faces (if any) that are part of the issue.

Import detail

```
from ansys.geometry.core.tools.check_geometry import GeometryIssue
```

Property detail

property `GeometryIssue.message_type: str`

The type of the message (warning, error, info).

property `GeometryIssue.message_id: str`

The identifier for the message.

property `GeometryIssue.message: str`

The content of the message.

property `GeometryIssue.edges: list[str]`

The List of edges (if any) that are part of the issue.

property `GeometryIssue.faces: list[str]`

The List of faces (if any) that are part of the issue.

InspectResult

class `ansys.geometry.core.tools.check_geometry.InspectResult`(*grpc_client*: `ansys.geometry.core.connection.client.GrpcClient`,
body: `ansys.geometry.core.designer.body.Body`,
issues: `list[GeometryIssue]`)

Provides the result of the inspect geometry operation.

Overview

Methods

<i>repair</i>	Repair the problem area.
---------------	--------------------------

Properties

<i>body</i>	The body for which issues are found.
<i>issues</i>	The list of issues for the body.

Import detail

```
from ansys.geometry.core.tools.check_geometry import InspectResult
```

Property detail

property `InspectResult.body: ansys.geometry.core.designer.body.Body`

The body for which issues are found.

property `InspectResult.issues: list[GeometryIssue]`

The list of issues for the body.

Method detail

`InspectResult.repair()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Repair the problem area.

Returns

RepairToolMessage

Message containing created and/or modified bodies.

Description

Module for repair tool message.

The `measurement_tools.py` module

Summary

Classes

<code>Gap</code>	Represents a gap between two bodies.
<code>MeasurementTools</code>	Measurement tools for PyAnsys Geometry.

Gap

```
class ansys.geometry.core.tools.measurement_tools.Gap(distance: an-  
sys.geometry.core.misc.measurements.Distance)
```

Represents a gap between two bodies.

Parameters

`distance`

[Distance] Distance between two sides of the gap.

Overview

Properties

<code>distance</code>	Returns the closest distance between two bodies.
-----------------------	--

Import detail

```
from ansys.geometry.core.tools.measurement_tools import Gap
```

Property detail

```
property Gap.distance: ansys.geometry.core.misc.measurements.Distance
```

Returns the closest distance between two bodies.

MeasurementTools

```
class ansys.geometry.core.tools.measurement_tools.MeasurementTools(grpc_client: an-  
sys.geometry.core.connection.GrpcClient)
```

Measurement tools for PyAnsys Geometry.

Parameters

`grpc_client`

[GrpcClient] gRPC client to use for the measurement tools.

Overview

Methods

<code>min_distance_between_objects</code>	Find the gap between two bodies.
---	----------------------------------

Import detail

```
from ansys.geometry.core.tools.measurement_tools import MeasurementTools
```

Method detail

`MeasurementTools.min_distance_between_objects`(*object1*: `ansys.geometry.core.designer.body.Body` | `ansys.geometry.core.designer.face.Face` | `ansys.geometry.core.designer.edge.Edge`, *object2*: `ansys.geometry.core.designer.body.Body` | `ansys.geometry.core.designer.face.Face` | `ansys.geometry.core.designer.edge.Edge`) → *Gap*

Find the gap between two bodies.

Parameters

object1

[Union[Body, Face, Edge]] First object to measure the gap.

object2

[Union[Body, Face, Edge]] Second object to measure the gap.

Returns

Gap

Gap between two bodies.

Description

Provides tools for measurement.

The `prepare_tools.py` module

Summary

Classes

<code>PrepareTools</code>	Prepare tools for PyAnsys Geometry.
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PrepareTools

```
class ansys.geometry.core.tools.prepare_tools.PrepareTools(grpc_client: ansys.geometry.core.connection.GrpcClient)
```

Prepare tools for PyAnsys Geometry.

Parameters

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

Overview

Methods

<code>extract_volume_from_faces</code>	Extract a volume from input faces.
<code>extract_volume_from_edge_loops</code>	Extract a volume from input edge loops.
<code>share_topology</code>	Share topology between the chosen bodies.
<code>enhanced_share_topology</code>	Share topology between the chosen bodies.

Import detail

```
from ansys.geometry.core.tools.prepare_tools import PrepareTools
```

Method detail

`PrepareTools.extract_volume_from_faces`(*sealing_faces*: *list*[*ansys.geometry.core.designer.face.Face*], *inside_faces*: *list*[*ansys.geometry.core.designer.face.Face*]) → *list*[*ansys.geometry.core.designer.body.Body*]

Extract a volume from input faces.

Creates a volume (typically a flow volume) from a list of faces that seal the volume and one or more faces that define the wetted surface (inside faces of the solid).

Parameters

sealing_faces

[*list*[*Face*]] List of faces that seal the volume.

inside_faces

[*list*[*Face*]] List of faces that define the interior of the solid.

Returns

list[*Body*]

List of created bodies.

`PrepareTools.extract_volume_from_edge_loops`(*sealing_edges*: *list*[*ansys.geometry.core.designer.edge.Edge*], *inside_faces*: *list*[*ansys.geometry.core.designer.face.Face*] = *None*) → *list*[*ansys.geometry.core.designer.body.Body*]

Extract a volume from input edge loops.

Creates a volume (typically a flow volume) from a list of edge loops that seal the volume. and one or more faces that define the wetted surface (inside faces of the solid).

Parameters

sealing_edges

[*list*[*Edge*]] List of faces that seal the volume.

inside_faces

[*list*[*Face*], optional] List of faces that define the interior of the solid (not always necessary).

Returns

list[*Body*]

List of created bodies.

PrepareTools.**share_topology**(*bodies*: list[ansys.geometry.core.designer.body.Body], *tol*:
ansys.geometry.core.typing.Real = 0.0, *preserve_instances*: *bool = False*) →
 bool

Share topology between the chosen bodies.

Parameters

bodies

[list[Body]] List of bodies to share topology between.

tol

[Real] Maximum distance between bodies.

preserve_instances

[bool] Whether instances are preserved.

Returns

bool

True if successful, False if failed.

PrepareTools.**enhanced_share_topology**(*bodies*: list[ansys.geometry.core.designer.body.Body], *tol*:
ansys.geometry.core.typing.Real = 0.0, *preserve_instances*: *bool = False*) →
ansys.geometry.core.tools.repair_tool_message.RepairToolMessage

Share topology between the chosen bodies.

Parameters

bodies

[list[Body]] List of bodies to share topology between.

tol

[Real] Maximum distance between bodies.

preserve_instances

[bool] Whether instances are preserved.

Returns

RepairToolMessage

Message containing number of problem areas found/fixes, created and/or modified bodies.

Description

Provides tools for preparing geometry for use with simulation.

The `problem_areas.py` module

Summary

Classes

<i>ProblemArea</i>	Represents problem areas.
<i>DuplicateFaceProblemAreas</i>	Provides duplicate face problem area definition.
<i>MissingFaceProblemAreas</i>	Provides missing face problem area definition.
<i>InexactEdgeProblemAreas</i>	Represents an inexact edge problem area with unique identifier and associated edges.
<i>ExtraEdgeProblemAreas</i>	Represents a extra edge problem area with unique identifier and associated edges.
<i>ShortEdgeProblemAreas</i>	Represents a short edge problem area with a unique identifier and associated edges.
<i>SmallFaceProblemAreas</i>	Represents a small face problem area with a unique identifier and associated faces.
<i>SplitEdgeProblemAreas</i>	Represents a split edge problem area with unique identifier and associated edges.
<i>StitchFaceProblemAreas</i>	Represents a stitch face problem area with unique identifier and associated faces.
<i>UnsimplifiedFaceProblemAreas</i>	Represents an unsimplified face problem area with unique identifier and associated faces.
<i>InterferenceProblemAreas</i>	Represents an interference problem area with a unique identifier and associated bodies.

ProblemArea

class `ansys.geometry.core.tools.problem_areas.ProblemArea`(*id*: *str*, *grpc_client*: *ansys.geometry.core.connection.GrpcClient*)

Represents problem areas.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

Overview

Abstract methods

fix Fix problem area.

Properties

id The id of the problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import ProblemArea
```

Property detail

property ProblemArea.**id**: **str**

The id of the problem area.

Method detail

abstract ProblemArea.**fix**()

Fix problem area.

DuplicateFaceProblemAreas

```
class ansys.geometry.core.tools.problem_areas.DuplicateFaceProblemAreas(id: str, grpc_client:
                                                                    an-
                                                                    sys.geometry.core.connection.GrpcClient,
                                                                    faces:
                                                                    list[ansys.geometry.core.designer.face.Face])
```

Bases: ProblemArea

Provides duplicate face problem area definition.

Represents a duplicate face problem area with unique identifier and associated faces.

Parameters

id

[**str**] Server-defined ID for the problem area.

grpc_client

[**GrpcClient**] Active supporting geometry service instance for design modeling.

faces

[**list**[**Face**]] List of faces associated with the design.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>faces</i> The list of faces connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import DuplicateFaceProblemAreas
```

Property detail

property DuplicateFaceProblemAreas.**faces**: **list**[**ansys.geometry.core.designer.face.Face**]

The list of faces connected to this problem area.

Method detail

`DuplicateFaceProblemAreas.fix()` → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

MissingFaceProblemAreas

class `ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas` (*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: list[ansys.geometry.core.designer.edge.Edge]*)

Bases: `ProblemArea`

Provides missing face problem area definition.

Parameters

id

[`str`] Server-defined ID for the problem area.

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

edges

[`list[Edge]`] List of edges associated with the design.

Overview

Methods

<code>fix</code>	Fix the problem area.
------------------	-----------------------

Properties

<code>edges</code>	The list of edges connected to this problem area.
--------------------	---

Import detail

```
from ansys.geometry.core.tools.problem_areas import MissingFaceProblemAreas
```

Property detail

property `MissingFaceProblemAreas.edges`: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

MissingFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: *RepairToolMessage*

Message containing created and/or modified bodies.

InexactEdgeProblemAreas

class *ansys.geometry.core.tools.problem_areas.InexactEdgeProblemAreas*(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: list[ansys.geometry.core.designer.edge.Edge]*)

Bases: *ProblemArea*

Represents an inexact edge problem area with unique identifier and associated edges.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

edges

[*list*[*Edge*]] List of edges associated with the design.

Overview

Methods

fix Fix the problem area.

Properties

edges The list of edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import InexactEdgeProblemAreas
```

Property detail

property *InexactEdgeProblemAreas.edges:* *list[ansys.geometry.core.designer.edge.Edge]*

The list of edges connected to this problem area.

Method detail

`InexactEdgeProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

ExtraEdgeProblemAreas

`class` `ansys.geometry.core.tools.problem_areas.ExtraEdgeProblemAreas`(`id`: `str`, `grpc_client`: `ansys.geometry.core.connection.GrpcClient`,
`edges`:
`list`[`ansys.geometry.core.designer.edge.Edge`])

Bases: `ProblemArea`

Represents a extra edge problem area with unique identifier and associated edges.

Parameters

id

[`str`] Server-defined ID for the problem area.

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

edges

[`list`[`Edge`]] List of edges associated with the design.

Overview

Methods

<code>fix</code>	Fix the problem area.
------------------	-----------------------

Properties

<code>edges</code>	The list of edges connected to this problem area.
--------------------	---

Import detail

```
from ansys.geometry.core.tools.problem_areas import ExtraEdgeProblemAreas
```

Property detail

property `ExtraEdgeProblemAreas.edges`: `list`[`ansys.geometry.core.designer.edge.Edge`]

The list of edges connected to this problem area.

Method detail

ExtraEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: *RepairToolMessage*

Message containing created and/or modified bodies.

ShortEdgeProblemAreas

class *ansys.geometry.core.tools.problem_areas.ShortEdgeProblemAreas*(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: list[ansys.geometry.core.designer.edge.Edge]*)

Bases: *ProblemArea*

Represents a short edge problem area with a unique identifier and associated edges.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

edges

[*list*[*Edge*]] List of edges associated with the design.

Overview

Methods

fix Fix the problem area.

Properties

edges The list of edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import ShortEdgeProblemAreas
```

Property detail

property *ShortEdgeProblemAreas.edges:* *list[ansys.geometry.core.designer.edge.Edge]*

The list of edges connected to this problem area.

Method detail

ShortEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: *RepairToolMessage*

Message containing created and/or modified bodies.

SmallFaceProblemAreas

class *ansys.geometry.core.tools.problem_areas.SmallFaceProblemAreas*(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, faces: list[ansys.geometry.core.designer.face.Face]*)

Bases: *ProblemArea*

Represents a small face problem area with a unique identifier and associated faces.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

faces

[*list*[*Face*]] List of edges associated with the design.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>faces</i> The list of faces connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import SmallFaceProblemAreas
```

Property detail

property *SmallFaceProblemAreas.faces:* *list[ansys.geometry.core.designer.face.Face]*

The list of faces connected to this problem area.

Method detail

`SmallFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

SplitEdgeProblemAreas

`class ansys.geometry.core.tools.problem_areas.SplitEdgeProblemAreas(id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: list[ansys.geometry.core.designer.edge.Edge])`

Bases: `ProblemArea`

Represents a split edge problem area with unique identifier and associated edges.

Parameters

id

[str] Server-defined ID for the problem area.

grpc_client

[GrpcClient] Active supporting geometry service instance for design modeling.

edges

[list[Edge]] List of edges associated with the design.

Overview

Methods

`fix` Fix the problem area.

Properties

`edges` The list of edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import SplitEdgeProblemAreas
```

Property detail

property `SplitEdgeProblemAreas.edges`: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

`SplitEdgeProblemAreas.fix()` → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

StitchFaceProblemAreas

class `ansys.geometry.core.tools.problem_areas.StitchFaceProblemAreas`(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, bodies: list[ansys.geometry.core.designer.body.Body]*

Bases: `ProblemArea`

Represents a stitch face problem area with unique identifier and associated faces.

Parameters

id

[`str`] Server-defined ID for the problem area.

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

bodies

[`list`[`Body`]] List of bodies associated with the design.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>bodies</i> The list of bodies connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import StitchFaceProblemAreas
```

Property detail

property `StitchFaceProblemAreas.bodies`: `list[ansys.geometry.core.designer.body.Body]`

The list of bodies connected to this problem area.

Method detail

`StitchFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

UnsimplifiedFaceProblemAreas

```
class ansys.geometry.core.tools.problem_areas.UnsimplifiedFaceProblemAreas(id: str,
                                                                           grpc_client: an-
                                                                           sys.geometry.core.connection.GrpcC
                                                                           faces:
                                                                           list[ansys.geometry.core.designer.face
```

Bases: `ProblemArea`

Represents a unsimplified face problem area with unique identifier and associated faces.

Parameters

id

[`str`] Server-defined ID for the problem area.

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

faces

[`list`[`Face`]] List of faces associated with the design.

Overview

Methods

`fix` Fix the problem area.

Properties

`faces` The list of faces connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import UnsimplifiedFaceProblemAreas
```

Property detail

property `UnsimplifiedFaceProblemAreas.faces`: `list[ansys.geometry.core.designer.face.Face]`

The list of faces connected to this problem area.

Method detail

UnsimplifiedFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: **RepairToolMessage**

Message containing created and/or modified bodies.

InterferenceProblemAreas

class *ansys.geometry.core.tools.problem_areas*.**InterferenceProblemAreas**(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, bodies: list[ansys.geometry.core.designer.body.Body]*)

Bases: *ProblemArea*

Represents an interference problem area with a unique identifier and associated bodies.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

bodies

[*list*[*Body*]] List of bodies in the problem area.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>bodies</i> The list of the bodies connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import InterferenceProblemAreas
```

Property detail

property *InterferenceProblemAreas*.**bodies:** *list[ansys.geometry.core.designer.body.Body]*

The list of the bodies connected to this problem area.

Method detail

`InterferenceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: RepairToolMessage

Message containing created and/or modified bodies.

Notes

The current implementation does not properly track changes. The list of created and modified bodies are empty.

Description

The problem area definition.

The repair_tool_message.py module

Summary

Classes

<code>RepairToolMessage</code>	Provides return message for the repair tool methods.
--------------------------------	--

RepairToolMessage

```
class ansys.geometry.core.tools.repair_tool_message.RepairToolMessage(success: bool,
                                                                    created_bodies: list[str],
                                                                    modified_bodies:
                                                                    list[str], found: int = -1,
                                                                    repaired: int = -1)
```

Provides return message for the repair tool methods.

Overview

Properties

<code>success</code>	The success of the repair operation.
<code>created_bodies</code>	The list of the created bodies after the repair operation.
<code>modified_bodies</code>	The list of the modified bodies after the repair operation.
<code>found</code>	Number of problem areas found for the repair operation.
<code>repaired</code>	Number of problem areas repaired during the repair operation.

Import detail

```
from ansys.geometry.core.tools.repair_tool_message import RepairToolMessage
```


Property detail

property `RepairToolMessage.success`: `bool`

The success of the repair operation.

property `RepairToolMessage.created_bodies`: `list[str]`

The list of the created bodies after the repair operation.

property `RepairToolMessage.modified_bodies`: `list[str]`

The list of the modified bodies after the repair operation.

property `RepairToolMessage.found`: `int`

Number of problem areas found for the repair operation.

property `RepairToolMessage.repaired`: `int`

Number of problem areas repaired during the repair operation.

Description

Module for repair tool message.

The `repair_tools.py` module

Summary

Classes

<code>RepairTools</code> Repair tools for PyAnsys Geometry.

RepairTools

```
class ansys.geometry.core.tools.repair_tools.RepairTools(grpc_client: an-  
sys.geometry.core.connection.GrpcClient,  
modeler:  
ansys.geometry.core.modeler.Modeler)
```

Repair tools for PyAnsys Geometry.

Overview

Methods

<code>find_split_edges</code>	Find split edges in the given list of bodies.
<code>find_extra_edges</code>	Find the extra edges in the given list of bodies.
<code>find_inexact_edges</code>	Find inexact edges in the given list of bodies.
<code>find_short_edges</code>	Find the short edge problem areas.
<code>find_duplicate_faces</code>	Find the duplicate face problem areas.
<code>find_missing_faces</code>	Find the missing faces.
<code>find_small_faces</code>	Find the small face problem areas.
<code>find_stitch_faces</code>	Return the list of stitch face problem areas.
<code>find_simplify</code>	Detect faces in a body that can be simplified.
<code>find_interferences</code>	Find the interference problem areas.
<code>find_and_fix_short_edges</code>	Find and fix the short edge problem areas.
<code>find_and_fix_extra_edges</code>	Find and fix the extra edge problem areas.
<code>find_and_fix_split_edges</code>	Find and fix the split edge problem areas.
<code>find_and_fix_simplify</code>	Find and simplify the provided geometry.
<code>inspect_geometry</code>	Return a list of geometry issues organized by body.
<code>repair_geometry</code>	Attempt to repair the geometry for the given bodies.

Import detail

```
from ansys.geometry.core.tools.repair_tools import RepairTools
```

Method detail

`RepairTools.find_split_edges`(*bodies*: *list*[`ansys.geometry.core.designer.body.Body`], *angle*: `ansys.geometry.core.typing.Real = 0.0`, *length*: `ansys.geometry.core.typing.Real = 0.0`) → *list*[`ansys.geometry.core.tools.problem_areas.SplitEdgeProblemAreas`]

Find split edges in the given list of bodies.

This method finds the split edge problem areas and returns a list of split edge problem areas objects.

Parameters

bodies

[*list*[`Body`]] List of bodies that split edges are investigated on.

angle

[`Real`] The maximum angle between edges.

length

[`Real`] The maximum length of the edges.

Returns

list[`SplitEdgeProblemAreas`]

List of objects representing split edge problem areas.

`RepairTools.find_extra_edges`(*bodies*: *list*[`ansys.geometry.core.designer.body.Body`]) → *list*[`ansys.geometry.core.tools.problem_areas.ExtraEdgeProblemAreas`]

Find the extra edges in the given list of bodies.

This method find the extra edge problem areas and returns a list of extra edge problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that extra edges are investigated on.

Returns**list[ExtraEdgeProblemArea]**

List of objects representing extra edge problem areas.

RepairTools.**find_inexact_edges**(*bodies*: list[ansys.geometry.core.designer.body.Body]) →
list[ansys.geometry.core.tools.problem_areas.InexactEdgeProblemAreas]

Find inexact edges in the given list of bodies.

This method find the inexact edge problem areas and returns a list of inexact edge problem areas objects.

Parameters**bodies**

[list[Body]] List of bodies that inexact edges are investigated on.

Returns**list[InExactEdgeProblemArea]**

List of objects representing inexact edge problem areas.

RepairTools.**find_short_edges**(*bodies*: list[ansys.geometry.core.designer.body.Body], *length*:
ansys.geometry.core.typing.Real = 0.0) →
list[ansys.geometry.core.tools.problem_areas.ShortEdgeProblemAreas]

Find the short edge problem areas.

This method finds the short edge problem areas and returns a list of these objects.

Parameters**bodies**

[list[Body]] List of bodies that short edges are investigated on.

Returns**list[ShortEdgeProblemAreas]**

List of objects representing short edge problem areas.

RepairTools.**find_duplicate_faces**(*bodies*: list[ansys.geometry.core.designer.body.Body]) →
list[ansys.geometry.core.tools.problem_areas.DuplicateFaceProblemAreas]

Find the duplicate face problem areas.

This method finds the duplicate face problem areas and returns a list of duplicate face problem areas objects.

Parameters**bodies**

[list[Body]] List of bodies that duplicate faces are investigated on.

Returns**list[DuplicateFaceProblemAreas]**

List of objects representing duplicate face problem areas.

RepairTools.**find_missing_faces**(*bodies*: list[ansys.geometry.core.designer.body.Body]) →
list[ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas]

Find the missing faces.

This method find the missing face problem areas and returns a list of missing face problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that missing faces are investigated on.

Returns**list[MissingFaceProblemAreas]**

List of objects representing missing face problem areas.

RepairTools.**find_small_faces**(*bodies: list[ansys.geometry.core.designer.body.Body]*) →
list[ansys.geometry.core.tools.problem_areas.SmallFaceProblemAreas]

Find the small face problem areas.

This method finds and returns a list of ids of small face problem areas objects.

Parameters**bodies**

[list[Body]] List of bodies that small faces are investigated on.

Returns**list[SmallFaceProblemAreas]**

List of objects representing small face problem areas.

RepairTools.**find_stitch_faces**(*bodies: list[ansys.geometry.core.designer.body.Body]*) →
list[ansys.geometry.core.tools.problem_areas.StitchFaceProblemAreas]

Return the list of stitch face problem areas.

This method find the stitch face problem areas and returns a list of ids of stitch face problem areas objects.

Parameters**bodies**

[list[Body]] List of bodies that stitchable faces are investigated on.

Returns**list[StitchFaceProblemAreas]**

List of objects representing stitch face problem areas.

RepairTools.**find_simplify**(*bodies: list[ansys.geometry.core.designer.body.Body]*) →
list[ansys.geometry.core.tools.problem_areas.UnsimplifiedFaceProblemAreas]

Detect faces in a body that can be simplified.

Parameters**bodies**

[list[Body]] List of bodies to search.

Returns**list[UnsimplifiedFaceProblemAreas]**

List of objects representing unsimplified face problem areas.

RepairTools.**find_interferences**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *cut_smaller_body: bool = False*) →
list[ansys.geometry.core.tools.problem_areas.InterferenceProblemAreas]

Find the interference problem areas.

Parameters**bodies**

[list[Body]] List of bodies that small faces are investigated on.

cut_smaller_body

[bool, optional] Whether to cut the smaller body if an interference is found. By default, False.

Returns

list[InterferenceProblemAreas]

List of objects representing interference problem areas.

Notes

This method finds and returns a list of ids of interference problem areas objects.

`RepairTools.find_and_fix_short_edges(bodies: list[ansys.geometry.core.designer.body.Body], length: ansys.geometry.core.typing.Real = 0.0, comprehensive_result: bool = False) → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Find and fix the short edge problem areas.

Parameters**bodies**

[list[Body]] List of bodies that short edges are investigated on.

length

[Real, optional] The maximum length of the edges. By default, 0.0.

comprehensive_result

[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixes, created and/or modified bodies.

Notes

This method finds the short edges in the bodies and fixes them.

`RepairTools.find_and_fix_extra_edges(bodies: list[ansys.geometry.core.designer.body.Body], comprehensive_result: bool = False) → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Find and fix the extra edge problem areas.

Parameters**bodies**

[list[Body]] List of bodies that short edges are investigated on.

length

[Real] The maximum length of the edges.

comprehensive_result

[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixes, created and/or modified bodies.

Notes

This method finds the extra edges in the bodies and fixes them.

`RepairTools.find_and_fix_split_edges`(*bodies*: *list*[*ansys.geometry.core.designer.body.Body*], *angle*: *ansys.geometry.core.typing.Real* = 0.0, *length*: *ansys.geometry.core.typing.Real* = 0.0, *comprehensive_result*: *bool* = *False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and fix the split edge problem areas.

Parameters

bodies

[*list*[*Body*]] List of bodies that split edges are investigated on.

angle

[*Real*, *optional*] The maximum angle between edges. By default, 0.0.

length

[*Real*, *optional*] The maximum length of the edges. By default, 0.0.

comprehensive_result

[*bool*, *optional*] Whether to fix all problem areas individually. By default, *False*.

Returns

RepairToolMessage

Message containing number of problem areas found/fixed, created and/or modified bodies.

Notes

This method finds the extra edges in the bodies and fixes them.

`RepairTools.find_and_fix_simplify`(*bodies*: *list*[*ansys.geometry.core.designer.body.Body*], *comprehensive_result*: *bool* = *False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and simplify the provided geometry.

Parameters

bodies

[*list*[*Body*]] List of bodies to be simplified.

comprehensive_result

[*bool*, *optional*] Whether to fix all problem areas individually. By default, *False*.

Returns

RepairToolMessage

Message containing number of problem areas found/fixed, created and/or modified bodies.

Notes

This method simplifies the provided geometry.

`RepairTools.inspect_geometry`(*bodies*: *list*[*ansys.geometry.core.designer.body.Body*] = *None*) → *list*[*ansys.geometry.core.tools.check_geometry.InspectResult*]

Return a list of geometry issues organized by body.

This method inspects the geometry and returns a list of the issues grouped by the body where they are found.

Parameters

bodies

[list[Body]] List of bodies to inspect the geometry for. All bodies are inspected if the argument is not given.

Returns

list[IssuesByBody]

List of objects representing geometry issues and the bodies where issues are found.

RepairTools.**repair_geometry**(*bodies: list[ansys.geometry.core.designer.body.Body] = None*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Attempt to repair the geometry for the given bodies.

This method inspects the geometry for the given bodies and attempts to repair them.

Parameters

bodies

[list[Body]] List of bodies where to attempt to repair the geometry. All bodies are repaired if the argument is not given.

Returns

RepairToolMessage

Message containing success of the operation.

Description

Provides tools for repairing bodies.

The unsupported.py module

Summary

Classes

<i>UnsupportedCommands</i>	Provides unsupported commands for PyAnsys Geometry.
----------------------------	---

Enums

<i>PersistentIdType</i>	Type of persistent id.
-------------------------	------------------------

UnsupportedCommands

class ansys.geometry.core.tools.unsupported.**UnsupportedCommands**(*grpc_client: ansys.geometry.core.connection.GrpcClient, modeler: ansys.geometry.core.modeler.Modeler*)

Provides unsupported commands for PyAnsys Geometry.

Parameters

grpc_client

[GrpcClient] gRPC client to use for the geometry commands.

modeler

[Modeler] Modeler instance to use for the geometry commands.

Overview

Methods

<code>set_export_id</code>	Set the persistent id for the moniker.
<code>get_body_occurrences_from_import_id</code>	Get all body occurrences whose master has the given import id.
<code>get_face_occurrences_from_import_id</code>	Get all face occurrences whose master has the given import id.
<code>get_edge_occurrences_from_import_id</code>	Get all edge occurrences whose master has the given import id.

Import detail

```
from ansys.geometry.core.tools.unsupported import UnsupportedCommands
```

Method detail

`UnsupportedCommands.set_export_id(moniker: str, id_type: PersistentIdType, value: str) → None`

Set the persistent id for the moniker.

Parameters

moniker

[str] Moniker to set the id for.

id_type

[PersistentIdType] Type of id.

value

[str] Id to set.

`UnsupportedCommands.get_body_occurrences_from_import_id(import_id: str, id_type: PersistentIdType)`
→

`list[ansys.geometry.core.designer.body.Body]`

Get all body occurrences whose master has the given import id.

Parameters

import_id

[str] Persistent id

id_type

[PersistentIdType] Type of id

Returns

`list[Body]`

List of body occurrences.

`UnsupportedCommands.get_face_occurrences_from_import_id(import_id: str, id_type: PersistentIdType)`
→

`list[ansys.geometry.core.designer.face.Face]`

Get all face occurrences whose master has the given import id.

Parameters

import_id

[str] Persistent id.

id_type

[PersistentIdType] Type of id.

Returns

`list[Face]`
 List of face occurrences.

`UnsupportedCommands.get_edge_occurrences_from_import_id(import_id: str, id_type: PersistentIdType)`
 →
`list[ansys.geometry.core.designer.edge.Edge]`

Get all edge occurrences whose master has the given import id.

Parameters

import_id
`[str]` Persistent id.

id_type
`[PersistentIdType]` Type of id.

Returns

`list[Edge]`
 List of edge occurrences.

PersistentIdType

`class ansys.geometry.core.tools.unsupported.PersistentIdType(*args, **kwargs)`

Bases: `enum.Enum`

Type of persistent id.

Overview

Attributes

<code>PNAME</code>
<code>PRIME_ID</code>

Import detail

```
from ansys.geometry.core.tools.unsupported import PersistentIdType
```

Attribute detail

`PersistentIdType.PNAME = 1`
`PersistentIdType.PRIME_ID = 700`

Description

Unsupported functions for the PyAnsys Geometry library.

Description

PyAnsys Geometry tools subpackage.

The `errors.py` module

Summary

Exceptions

<code>GeometryRuntimeError</code>	Provides error message when Geometry service passes a runtime error.
<code>GeometryExitedError</code>	Provides error message to raise when Geometry service has exited.

Functions

<code>handler</code>	Pass signal to the custom interrupt handler.
<code>protect_grpc</code>	Capture gRPC exceptions and raise a more succinct error message.

Constants

<code>SIGINT_TRACKER</code>

`GeometryRuntimeError`

exception `ansys.geometry.core.errors.GeometryRuntimeError`

Bases: `RuntimeError`

Provides error message when Geometry service passes a runtime error.

Import detail

```
from ansys.geometry.core.errors import GeometryRuntimeError
```

`GeometryExitedError`

exception `ansys.geometry.core.errors.GeometryExitedError` (`msg='Geometry service has exited.'`)

Bases: `RuntimeError`

Provides error message to raise when Geometry service has exited.

Parameters

msg

[`str`, default: "Geometry service has exited."] Message to raise.

Import detail

```
from ansys.geometry.core.errors import GeometryExitedError
```

Description

Provides PyAnsys Geometry-specific errors.

Module detail

`errors.handler(sig, frame)`

Pass signal to the custom interrupt handler.

`errors.protect_grpc(func)`

Capture gRPC exceptions and raise a more succinct error message.

This method captures the `KeyboardInterrupt` exception to avoid segfaulting the Geometry service.

While this works some of the time, it does not work all of the time. For some reason, gRPC still captures SIGINT.

`errors.SIGINT_TRACKER = []`

The logger.py module

Summary

Classes

<code>PyGeometryCustomAdapter</code>	Keeps the reference to the Geometry service instance name dynamic.
<code>PyGeometryPercentStyle</code>	Provides a common messaging style.
<code>PyGeometryFormatter</code>	Provides a <code>Formatter</code> class for overwriting default format styles.
<code>InstanceFilter</code>	Ensures that the <code>instance_name</code> record always exists.
<code>Logger</code>	Provides the logger used for each PyAnsys Geometry session.

Functions

<code>addfile_handler</code>	Add a file handler to the input.
<code>add_stdout_handler</code>	Add a stdout handler to the logger.

Attributes

<code>string_to_loglevel</code>

Constants

<code>LOG_LEVEL</code>
<code>FILE_NAME</code>
<code>DEBUG</code>
<code>INFO</code>
<code>WARN</code>
<code>ERROR</code>
<code>CRITICAL</code>
<code>STDOUT_MSG_FORMAT</code>
<code>FILE_MSG_FORMAT</code>
<code>DEFAULT_STDOUT_HEADER</code>
<code>DEFAULT_FILE_HEADER</code>
<code>NEW_SESSION_HEADER</code>
<code>LOG</code>

PyGeometryCustomAdapter

class `ansys.geometry.core.logger.PyGeometryCustomAdapter`(*logger, extra=None*)

Bases: `logging.LoggerAdapter`

Keeps the reference to the Geometry service instance name dynamic.

If you use the standard approach, which is supplying *extra* input to the logger, you must input Geometry service instances each time you do a log.

Using adapters, you only need to specify the Geometry service instance that you are referring to once.

Overview

Methods

<code>process</code>	Process the logging message and keyword arguments passed in to
<code>log_to_file</code>	Add a file handler to the logger.
<code>log_to_stdout</code>	Add a standard output handler to the logger.
<code>setLevel</code>	Change the log level of the object and the attached handlers.

Attributes

<code>level</code>
<code>file_handler</code>
<code>stdout_handler</code>
<code>logger</code>
<code>std_out_handler</code>

Import detail

```
from ansys.geometry.core.logger import PyGeometryCustomAdapter
```

Attribute detail

`PyGeometryCustomAdapter.level = None`

`PyGeometryCustomAdapter.file_handler = None`

`PyGeometryCustomAdapter.stdout_handler = None`

`PyGeometryCustomAdapter.logger`

`PyGeometryCustomAdapter.std_out_handler`

Method detail

`PyGeometryCustomAdapter.process(msg, kwargs)`

Process the logging message and keyword arguments passed in to a logging call to insert contextual information. You can either manipulate the message itself, the keyword args or both. Return the message and kwargs modified (or not) to suit your needs.

Normally, you'll only need to override this one method in a `LoggerAdapter` subclass for your specific needs.

`PyGeometryCustomAdapter.log_to_file(filename: str = FILE_NAME, level: int = LOG_LEVEL)`

Add a file handler to the logger.

Parameters

filename

[`str`, default: "pyansys-geometry.log"] Name of the file to write log messages to.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`PyGeometryCustomAdapter.log_to_stdout(level=LOG_LEVEL)`

Add a standard output handler to the logger.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`PyGeometryCustomAdapter.setLevel(level='DEBUG')`

Change the log level of the object and the attached handlers.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

PyGeometryPercentStyle

`class ansys.geometry.core.logger.PyGeometryPercentStyle(fmt, *, defaults=None)`

Bases: `logging.PercentStyle`

Provides a common messaging style.

Import detail

```
from ansys.geometry.core.logger import PyGeometryPercentStyle
```

PyGeometryFormatter

```
class ansys.geometry.core.logger.PyGeometryFormatter(fmt=STDOUT_MSG_FORMAT,
                                                    datefmt=None, style='%', validate=True,
                                                    defaults=None)
```

Bases: `logging.Formatter`

Provides a `Formatter` class for overwriting default format styles.

Import detail

```
from ansys.geometry.core.logger import PyGeometryFormatter
```

InstanceFilter

```
class ansys.geometry.core.logger.InstanceFilter(name="")
```

Bases: `logging.Filter`

Ensures that the `instance_name` record always exists.

Overview

Methods

<i>filter</i>	Ensure that the <code>instance_name</code> attribute is always present.
---------------	---

Import detail

```
from ansys.geometry.core.logger import InstanceFilter
```

Method detail

`InstanceFilter.filter(record)`

Ensure that the `instance_name` attribute is always present.

Logger

```
class ansys.geometry.core.logger.Logger(level=logging.DEBUG, to_file=False, to_stdout=True,
                                         filename=FILE_NAME)
```

Provides the logger used for each PyAnsys Geometry session.

This class allows you to add handlers to the logger to output messages to a file or to the standard output (stdout).

Parameters

level

[`int`, default: 10] Logging level to filter the message severity allowed in the logger. The default is 10, in which case the `logging.DEBUG` level is used.

to_file

[bool, default: False] Whether to write log messages to a file.

to_stdout

[bool, default: True] Whether to write log messages to the standard output.

filename

[str, default: "pyansys-geometry.log"] Name of the file to write log log messages to.

Examples

Demonstrate logger usage from the Modeler instance, which is automatically created when a Geometry service instance is created.

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler(loglevel="DEBUG")
>>> modeler._log.info("This is a useful message")
INFO - - <ipython-input-24-80df150fe31f> - <module> - This is LOG debug message.
```

Import the global PyAnsys Geometry logger and add a file output handler.

```
>>> import os
>>> from ansys.geometry.core import LOG
>>> file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
>>> LOG.log_to_file(file_path)
```

Overview

Methods

<i>log_to_file</i>	Add a file handler to the logger.
<i>log_to_stdout</i>	Add the standard output handler to the logger.
<i>setLevel</i>	Change the log level of the object and the attached handlers.
<i>add_child_logger</i>	Add a child logger to the main logger.
<i>add_instance_logger</i>	Add a logger for a Geometry service instance.
<i>add_handling_uncaught_exceptions</i>	Redirect the output of an exception to a logger.

Attributes

<i>file_handler</i>
<i>std_out_handler</i>
<i>logger</i>
<i>level</i>
<i>debug</i>
<i>info</i>
<i>warning</i>
<i>error</i>
<i>critical</i>
<i>log</i>

Special methods

<code>__getitem__</code>	Overload the access method by item for the <code>Logger</code> class.
--------------------------	---

Import detail

```
from ansys.geometry.core.logger import Logger
```

Attribute detail

`Logger.file_handler = None`

`Logger.std_out_handler = None`

`Logger.logger`

`Logger.level = 0`

`Logger.debug`

`Logger.info`

`Logger.warning`

`Logger.error`

`Logger.critical`

`Logger.log`

Method detail

`Logger.log_to_file(filename=FILE_NAME, level=LOG_LEVEL)`

Add a file handler to the logger.

Parameters

filename

[`str`, default: "pyansys-geometry.log"] Name of the file to write log messages to.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the logging.DEBUG level is used.

Examples

Write to the "pyansys-geometry.log" file in the current working directory:

```
>>> from ansys.geometry.core import LOG
>>> import os
>>> file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
>>> LOG.log_to_file(file_path)
```


`Logger.log_to_stdout(level=LOG_LEVEL)`

Add the standard output handler to the logger.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`Logger.setLevel(level='DEBUG')`

Change the log level of the object and the attached handlers.

`Logger.add_child_logger(suffix: str, level: str | None = None)`

Add a child logger to the main logger.

This logger is more general than an instance logger, which is designed to track the state of Geometry service instances.

If the logging level is in the arguments, a new logger with a reference to the `_global` logger handlers is created instead of a child logger.

Parameters

suffix

[`str`] Name of the child logger.

level

[`str`, default: `None`] Level of logging.

Returns

`logging.Logger`

Logger class.

`Logger.add_instance_logger(name: str, client_instance: ansys.geometry.core.connection.client.GrpcClient, level: int | None = None) → PyGeometryCustomAdapter`

Add a logger for a Geometry service instance.

The Geometry service instance logger is a logger with an adapter that adds contextual information such as the Geometry service instance name. This logger is returned, and you can use it to log events as a normal logger. It is stored in the `_instances` field.

Parameters

name

[`str`] Name for the new instance logger.

client_instance

[`GrpcClient`] Geometry service `GrpcClient` object, which should contain the `get_name` method.

level

[`int`, default: `None`] Level of logging.

Returns

`PyGeometryCustomAdapter`

Logger adapter customized to add Geometry service information to the logs. You can use this class to log events in the same way you would with the `Logger` class.

`Logger.__getitem__(key)`

Overload the access method by item for the `Logger` class.

`Logger.add_handling_uncaught_exceptions(logger)`

Redirect the output of an exception to a logger.

Parameters

logger
[str] Name of the logger.

Description

Provides a general framework for logging in PyAnsys Geometry.

This module is built on the [Logging facility for Python](#). It is not intended to replace the standard Python logging library but rather provide a way to interact between its logging class and PyAnsys Geometry.

The loggers used in this module include the name of the instance, which is intended to be unique. This name is printed in all active outputs and is used to track the different Geometry service instances.

Logger usage

Global logger

There is a global logger named `PyAnsys_Geometry_global` that is created when `ansys.geometry.core.__init__` is called. If you want to use this global logger, you must call it at the top of your module:

```
from ansys.geometry.core import LOG
```

You can rename this logger to avoid conflicts with other loggers (if any):

```
from ansys.geometry.core import LOG as logger
```

The default logging level of LOG is ERROR. You can change this level and output lower-level messages with this code:

```
LOG.logger.setLevel("DEBUG")
LOG.file_handler.setLevel("DEBUG") # If present.
LOG.stdout_handler.setLevel("DEBUG") # If present.
```

Alternatively, you can ensure that all the handlers are set to the input log level with this code:

```
LOG.setLevel("DEBUG")
```

This logger does not log to a file by default. If you want, you can add a file handler with this code:

```
import os

file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
LOG.log_to_file(file_path)
```

This also sets the logger to be redirected to this file. If you want to change the characteristics of this global logger from the beginning of the execution, you must edit the `__init__` file in the directory `ansys.geometry.core`.

To log using this logger, call the desired method as a normal logger with:

```
>>> import logging
>>> from ansys.geometry.core.logging import Logger
>>> LOG = Logger(level=logging.DEBUG, to_file=False, to_stdout=True)
>>> LOG.debug("This is LOG debug message.")
```

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```
DEBUG - - <ipython-input-24-80df150fe31f> - <module> - This is LOG debug message.
```

Instance logger

Every time an instance of the *Modeler* class is created, a logger is created and stored in `LOG._instances`. This field is a dictionary where the key is the name of the created logger.

These instance loggers inherit the `PyAnsys_Geometry_global` output handlers and logging level unless otherwise specified. The way this logger works is very similar to the global logger. If you want to add a file handler, you can use the `log_to_file()` method. If you want to change the log level, you can use the `setLevel()` method.

Here is an example of how you can use this logger:

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> modeler._log.info("This is a useful message")

INFO - GRPC_127.0.0.1:50056 - <...> - <module> - This is a useful message
```

Other loggers

You can create your own loggers using a Python logging library as you would do in any other script. There would be no conflicts between these loggers.

Module detail

`logger.addfile_handler(logger, filename=FILE_NAME, level=LOG_LEVEL, write_headers=False)`

Add a file handler to the input.

Parameters

logger

[`logging.Logger`] Logger to add the file handler to.

filename

[`str`, default: "pyansys-geometry.log"] Name of the output file.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

write_headers

[`bool`, default: `False`] Whether to write the headers to the file.

Returns

Logger

Logger or `logging.Logger` object.

`logger.add_stdout_handler(logger, level=LOG_LEVEL, write_headers=False)`

Add a stdout handler to the logger.

Parameters

logger

[`logging.Logger`] Logger to add the file handler to.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

write_headers

[`bool`, default: `False`] Whether to write headers to the file.

Returns**Logger**

Logger or `logging.Logger` object.

`logger.LOG_LEVEL = 10`

`logger.FILE_NAME = 'pyansys-geometry.log'`

`logger.DEBUG = 10`

`logger.INFO = 20`

`logger.WARN = 30`

`logger.ERROR = 40`

`logger.CRITICAL = 50`

`logger.STDOUT_MSG_FORMAT = '%(levelname)s - %(instance_name)s - %(module)s - %(funcName)s - %(message)s'`

`logger.FILE_MSG_FORMAT = '%(levelname)s - %(instance_name)s - %(module)s - %(funcName)s - %(message)s'`

`logger.DEFAULT_STDOUT_HEADER = Multiline-String`

```
"""
LEVEL - INSTANCE NAME - MODULE - FUNCTION - MESSAGE
"""
```

`logger.DEFAULT_FILE_HEADER = Multiline-String`

```
"""
LEVEL - INSTANCE NAME - MODULE - FUNCTION - MESSAGE
"""
```

`logger.NEW_SESSION_HEADER = Multiline-String`

```
"""
=====
NEW SESSION - Uninferable
===== """
```

`logger.LOG`

`logger.string_to_loglevel`

The `modeler.py` module

Summary

Classes

<i>Modeler</i> Provides for interacting with an open session of the Geometry service.

Modeler

```
class ansys.geometry.core.modeler.Modeler(host: str = DEFAULT_HOST, port: str | int =
    DEFAULT_PORT, channel: grpc.Channel | None = None,
    remote_instance:
    ansys.platform.instancemanagement.Instance | None = None,
    docker_instance: an-
    sys.geometry.core.connection.docker_instance.LocalDockerInstance
    | None = None, product_instance: an-
    sys.geometry.core.connection.product_instance.ProductInstance
    | None = None, timeout: ansys.geometry.core.typing.Real =
    120, logging_level: int = logging.INFO, logging_file:
    pathlib.Path | str | None = None, backend_type:
    ansys.geometry.core.connection.backend.BackendType | None
    = None)
```

Provides for interacting with an open session of the Geometry service.

Parameters

host

[str, default: DEFAULT_HOST] Host where the server is running.

port

[str | int, default: DEFAULT_PORT] Port number where the server is running.

channel

[Channel, default: None] gRPC channel for server communication.

remote_instance

[ansys.platform.instancemanagement.Instance, default: None] Corresponding remote instance when the Geometry service is launched using PyPIM. This instance is deleted when the `GrpcClient.close` method is called.

docker_instance

[LocalDockerInstance, default: None] Corresponding local Docker instance when the Geometry service is launched using the `launch_docker_modeler` method. This instance is deleted when the `GrpcClient.close` method is called.

product_instance

[ProductInstance, default: None] Corresponding local product instance when the product (Discovery or SpaceClaim) is launched through the `launch_modeler_with_geometry_service()`, `launch_modeler_with_discovery()` or the `launch_modeler_with_spaceclaim()` interface. This instance will be deleted when the `GrpcClient.close` method is called.

timeout

[Real, default: 120] Time in seconds for trying to achieve the connection.

logging_level

[int, default: INFO] Logging level to apply to the client.

logging_file

[*str*, Path, default: *None*] File to output the log to, if requested.

Overview

Methods

<i>create_design</i>	Initialize a new design with the connected client.
<i>get_active_design</i>	Get the active design on the modeler object.
<i>read_existing_design</i>	Read the existing design on the service with the connected client.
<i>close</i>	Access the client's close method.
<i>exit</i>	Access the client's close method.
<i>open_file</i>	Open a file.
<i>run_discovery_script_file</i>	Run a Discovery script file.
<i>get_service_logs</i>	Get the service logs.

Properties

<i>client</i>	Modeler instance client.
<i>design</i>	Retrieve the design within the modeler workspace.
<i>designs</i>	Retrieve the design within the modeler workspace.
<i>repair_tools</i>	Access to repair tools.
<i>prepare_tools</i>	Access to prepare tools.
<i>measurement_tools</i>	Access to measurement tools.
<i>geometry_commands</i>	Access to geometry commands.
<i>unsupported</i>	Access to unsupported commands.

Special methods

<code>__repr__</code>	Represent the modeler as a string.
-----------------------	------------------------------------

Import detail

```
from ansys.geometry.core.modeler import Modeler
```

Property detail

property `Modeler.client`: *ansys.geometry.core.connection.client.GrpcClient*
 Modeler instance client.

property `Modeler.design`: *ansys.geometry.core.designer.design.Design*
 Retrieve the design within the modeler workspace.

property `Modeler.designs`: `dict[str, ansys.geometry.core.designer.design.Design]`
 Retrieve the design within the modeler workspace.

Notes

This method is deprecated. Use the `design()` property instead.

property `Modeler.repair_tools`: *ansys.geometry.core.tools.repair_tools.RepairTools*

Access to repair tools.

property `Modeler.prepare_tools`: *ansys.geometry.core.tools.prepare_tools.PrepareTools*

Access to prepare tools.

property `Modeler.measurement_tools`:

ansys.geometry.core.tools.measurement_tools.MeasurementTools

Access to measurement tools.

property `Modeler.geometry_commands`:

ansys.geometry.core.designer.geometry_commands.GeometryCommands

Access to geometry commands.

property `Modeler.unsupported`: *ansys.geometry.core.tools.unsupported.UnsupportedCommands*

Access to unsupported commands.

Method detail

`Modeler.create_design(name: str) → ansys.geometry.core.designer.design.Design`

Initialize a new design with the connected client.

Parameters

name

[str] Name for the new design.

Returns

Design

Design object created on the server.

`Modeler.get_active_design(sync_with_backend: bool = True) → ansys.geometry.core.designer.design.Design`

Get the active design on the modeler object.

Parameters

sync_with_backend

[bool, default: True] Whether to sync the active design with the remote service. If set to False, the active design may be out-of-sync with the remote service. This is useful when the active design is known to be up-to-date.

Returns

Design

Design object already existing on the modeler.

`Modeler.read_existing_design() → ansys.geometry.core.designer.design.Design`

Read the existing design on the service with the connected client.

Returns

Design

Design object already existing on the server.

Modeler.close(*close_design: bool = True*) → None

Access the client's close method.

Parameters

close_design

[bool, default: True] Whether to close the design before closing the client.

Modeler.exit(*close_design: bool = True*) → None

Access the client's close method.

Parameters

close_design

[bool, default: True] Whether to close the design before closing the client.

Notes

This method is calling the same method as `close()`.

Modeler.open_file(*file_path: str | pathlib.Path, upload_to_server: bool = True, import_options: ansys.geometry.core.misc.options.ImportOptions = ImportOptions()*) → *ansys.geometry.core.designer.design.Design*

Open a file.

This method imports a design into the service. On Windows, .sdocx and HOOPS Exchange formats are supported. On Linux, only the .sdocx format is supported.

If the file is a shattered assembly with external references, the whole containing folder will need to be uploaded. Ensure proper folder structure in order to prevent the uploading of unnecessary files.

Parameters

file_path

[str, Path] Path of the file to open. The extension of the file must be included.

upload_to_server

[bool] True if the service is running on a remote machine. If service is running on the local machine, set to False, as there is no reason to upload the file.

import_options

[ImportOptions] Import options that toggle certain features when opening a file.

Returns

Design

Newly imported design.

Modeler.__repr__() → str

Represent the modeler as a string.

Modeler.run_discovery_script_file(*file_path: str | pathlib.Path, script_args: dict[str, str] | None = None, import_design: bool = False, api_version: int | str | ansys.geometry.core.connection.backend.ApiVersions = None*) → tuple[dict[str, str], *ansys.geometry.core.designer.design.Design* | None]

Run a Discovery script file.

Note

If arguments are passed to the script, they must be in the form of a dictionary. On the server side, the script will receive the arguments as a dictionary of strings, under the variable name `argsDict`. For example, if the script is called with the arguments `run_discovery_script_file(..., script_args = {"length": "20"}, ...)`, the script will receive the dictionary `argsDict` with the key-value pair `{"length": "20"}`.

Note

If an output is expected from the script, it will be returned as a dictionary of strings. The keys and values of the dictionary are the variables and their values that the script returns. However, it is necessary that the script creates a dictionary called `result` with the variables and their values that are expected to be returned. For example, if the script is expected to return the number of bodies in the design, the script should create a dictionary called `result` with the key-value pair `{"numBodies": numBodies}`, where `numBodies` is the number of bodies in the design.

The implied API version of the script should match the API version of the running Geometry Service. DMS API versions 24.1 and later are supported. DMS is a Windows-based modeling service that has been containerized to ease distribution, execution, and remotability operations.

Parameters**file_path**

[`str` | `Path`] Path of the file. The extension of the file must be included.

script_args

[`dict`[`str`, `str`], optional.] Arguments to pass to the script. By default, `None`.

import_design

[`bool`, optional.] Whether to refresh the current design from the service. When the script is expected to modify the existing design, set this to `True` to retrieve up-to-date design data. When this is set to `False` (default) and the script modifies the current design, the design may be out-of-sync. By default, `False`.

api_version

[`int` | `str` | `ApiVersions`, optional] The scripting API version to use. For example, version 24.1 can be passed as an integer 241, a string "241" or using the `ansys.geometry.core.connection.backend.ApiVersions` enum class. By default, `None`. When specified, the service will attempt to run the script with the specified API version. If the API version is not supported, the service will raise an error. If you are using Discovery or SpaceClaim, the product will determine the API version to use, so there is no need to specify this parameter.

Returns**dict**[`str`, `str`]

Values returned from the script.

Design, optional

Up-to-date current design. This is only returned if `import_design=True`.

Raises**GeometryRuntimeError**

If the Discovery script fails to run. Otherwise, assume that the script ran successfully.

Notes

The Ansys Geometry Service only supports scripts that are of the same version as the running service. Any `api_version` input will be ignored.

`Modeler.get_service_logs(all_logs: bool = False, dump_to_file: bool = False, logs_folder: str | pathlib.Path | None = None) → str | dict[str, str] | pathlib.Path`

Get the service logs.

Parameters

`all_logs`

[bool, default: False] Flag indicating whether all logs should be retrieved. By default, only the current logs are retrieved.

`dump_to_file`

[bool, default: False] Flag indicating whether the logs should be dumped to a file. By default, the logs are not dumped to a file.

`logs_folder`

[str, Path or None, default: None] Name of the folder where the logs should be dumped. This parameter is only used if the `dump_to_file` parameter is set to True.

Returns

`str`

Service logs as a string. This is returned if the `dump_to_file` parameter is set to False.

`dict[str, str]`

Dictionary containing the logs. The keys are the logs names, and the values are the logs as strings. This is returned if the `all_logs` parameter is set to True and the `dump_to_file` parameter is set to False.

`Path`

Path to the folder containing the logs (if the `all_logs` parameter is set to True) or the path to the log file (if only the current logs are retrieved). The `dump_to_file` parameter must be set to True.

Description

Provides for interacting with the Geometry service.

The typing.py module

Summary

Attributes

<code>Real</code>	Type used to refer to both integers and floats as possible values.
<code>RealSequence</code>	Type used to refer to Real types as a Sequence type.

Description

Provides typing of values for PyAnsys Geometry.

Module detail

`typing.Real`

Type used to refer to both integers and floats as possible values.

`typing.RealSequence`

Type used to refer to Real types as a Sequence type.

Notes

`numpy.ndarrays` are also accepted because they are the overlaying data structure behind most PyAnsys Geometry objects.

7.1.2 Description

PyAnsys Geometry is a Python wrapper for the Ansys Geometry service.

7.1.3 Module detail

`core.USE_SERVICE_COLORS: bool = False`

Global constant for checking whether to use service colors for plotting purposes. If set to False, the default colors will be used (speed gain).

`core.DISABLE_MULTIPLE_DESIGN_CHECK: bool = True`

Deprecated constant. Use `DISABLE_ACTIVE_DESIGN_CHECK` instead.

`core.DISABLE_ACTIVE_DESIGN_CHECK: bool = False`

Global constant for disabling the `ensure_design_is_active` check.

Only set this to false if you are sure you want to disable this check and your objects will always exist on the server side.

`core.DOCUMENTATION_BUILD: bool`

Global flag for the documentation to use the proper PyVista Jupyter backend.

`core.__version__`


PyAnsys Geometry version.

EXAMPLES

These examples demonstrate the behavior and usage of PyAnsys Geometry.

8.1 PyAnsys Geometry 101 examples

These examples demonstrate basic operations you can perform with PyAnsys Geometry.

 **Download this example**

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8.1.1 PyAnsys Geometry 101: Math

The `math` module is the foundation of PyAnsys Geometry. This module is built on top of `NumPy`, one of the most renowned mathematical Python libraries.

This example shows some of the main PyAnsys Geometry math objects and demonstrates why they are important prior to doing more exciting things in PyAnsys Geometry.

Perform required imports

Perform the required imports.

```
[1]: import numpy as np

from ansys.geometry.core.math import Plane, Point2D, Point3D, Vector2D, Vector3D, \
↳ UnitVector3D
```

Create points and vectors

Everything starts with `Point` and `Vector` objects, which can each be defined in a 2D or 3D form. These objects inherit from `NumPy`'s `ndarray`, providing them with enhanced functionalities. When creating these objects, you must remember to pass in the arguments as a list (that is, with brackets `[]`).

Create 2D and 3D point and vectors.

```
Point3D([x, y, z])
Point2D([x, y])
```

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```
Vector3D([x, y, z])
Vector2D([x, y])
```

You can perform standard mathematical operations on points and vectors.

Perform some standard operations on vectors.

```
[2]: vec_1 = Vector3D([1,0,0]) # x-vector
      vec_2 = Vector3D([0,1,0]) # y-vector

      print("Sum of vectors [1, 0, 0] + [0, 1, 0]:")
      print(vec_1 + vec_2) # sum

      print("\nDot product of vectors [1, 0, 0] * [0, 1, 0]:")
      print(vec_1 * vec_2) # dot

      print("\nCross product of vectors [1, 0, 0] % [0, 1, 0]:")
      print(vec_1 % vec_2) # cross

Sum of vectors [1, 0, 0] + [0, 1, 0]:
[1 1 0]

Dot product of vectors [1, 0, 0] * [0, 1, 0]:
0

Cross product of vectors [1, 0, 0] % [0, 1, 0]:
[0 0 1]
```

Create a vector from two points.

```
[3]: p1 = Point3D([12.4, 532.3, 89])
      p2 = Point3D([-5.7, -67.4, 46.6])

      vec_3 = Vector3D.from_points(p1, p2)
      vec_3

[3]: Vector3D([ -18.1, -599.7, -42.4])
```

Normalize a vector to create a unit vector, which is also known as a *direction*.

```
[4]: print("Magnitude of vec_3:")
      print(vec_3.magnitude)

      print("\nNormalized vec_3:")
      print(vec_3.normalize())

      print("\nNew magnitude:")
      print(vec_3.normalize().magnitude)

Magnitude of vec_3:
601.4694173438911

Normalized vec_3:
[-0.03009297 -0.99705818 -0.07049402]
```

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```
New magnitude:
1.0
```

Use the `UnitVector` class to automatically normalize the input for the unit vector.

```
[5]: uv = UnitVector3D([1,1,1])
      uv
[5]: UnitVector3D([0.57735027, 0.57735027, 0.57735027])
```

Perform a few more mathematical operations on vectors.

```
[6]: v1 = Vector3D([1, 0, 0])
      v2 = Vector3D([0, 1, 0])

      print("Vectors are perpendicular:")
      print(v1.is_perpendicular_to(v2))

      print("\nVectors are parallel:")
      print(v1.is_parallel_to(v2))

      print("\nVectors are opposite:")
      print(v1.is_opposite(v2))

      print("\nAngle between vectors:")
      print(v1.get_angle_between(v2))
      print(f"{np.pi / 2} == pi/2")
```

```
Vectors are perpendicular:
True

Vectors are parallel:
False

Vectors are opposite:
False

Angle between vectors:
1.5707963267948966 radian
1.5707963267948966 == pi/2
```

Create planes

Once you begin creating sketches and bodies, Plane objects become very important. A plane is defined by these items:

- An origin, which consists of a 3D point
- Two directions (`direction_x` and `direction_y`), which are both `UnitVector3D` objects

If no direction vectors are provided, the plane defaults to the XY plane.

Create two planes.

```
[7]: plane = Plane(Point3D([0,0,0])) # XY plane
```

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```
print("(1, 2, 0) is in XY plane:")
print(plane.is_point_contained(Point3D([1, 2, 0]))) # True

print("\n(0, 0, 5) is in XY plane:")
print(plane.is_point_contained(Point3D([0, 0, 5]))) # False
```

(1, 2, 0) is in XY plane:
True

(0, 0, 5) is in XY plane:
False

Perform parametric evaluations

PyAnsys Geometry implements parametric evaluations for some curves and surfaces.

Evaluate a sphere.

```
[8]: from ansys.geometry.core.shapes import Sphere, SphereEvaluation
      from ansys.geometry.core.math import Point3D
      from ansys.geometry.core.misc import Distance
```

```
sphere = Sphere(Point3D([0,0,0]), Distance(1)) # radius = 1
```

```
eval = sphere.project_point(Point3D([1,1,1]))
```

```
print("U Parameter:")
print(eval.parameter.u)
```

```
print("\nV Parameter:")
print(eval.parameter.v)
```

```
U Parameter:
0.7853981633974483
```

```
V Parameter:
0.6154797086703873
```

```
[9]: print("Point on the sphere:")
      eval.position
```

```
Point on the sphere:
```

```
[9]: Point3D([0.57735027, 0.57735027, 0.57735027])
```

```
[10]: print("Normal to the surface of the sphere at the evaluation position:")
       eval.normal
```

```
Normal to the surface of the sphere at the evaluation position:
```

```
[10]: UnitVector3D([0.57735027, 0.57735027, 0.57735027])
```

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8.1.2 PyAnsys Geometry 101: Units

To handle units inside the source code, PyAnsys Geometry uses [Pint](#), a third-party open source software that other PyAnsys libraries also use.

The following code examples show how to operate with units inside the PyAnsys Geometry codebase and create objects with different units.

Import units handler

The following line of code imports the units handler: `pint.util.UnitRegistry`. For more information on the `UnitRegistry` class in the `pint` API, see [Most important classes](#) in the `Pint` documentation.

```
[1]: from ansys.geometry.core.misc import UNITS
```

Create and work with Quantity objects

With the `UnitRegistry` object called `UNITS`, you can create `Quantity` objects. A `Quantity` object is simply a container class with two core elements:

- A number
- A unit

`Quantity` objects have convenience methods, including those for transforming to different units and comparing magnitudes, values, and units. For more information on the `Quantity` class in the `pint` API, see [Most important classes](#) in the `Pint` documentation. You can also step through this [Pint tutorial](#).

```
[2]: from pint import Quantity

a = Quantity(10, UNITS.mm)

print(f"Object a is a pint.Quantity: {a}")

print("Request its magnitude in different ways (accessor methods):")
print(f"Magnitude: {a.m}.")
print(f"Also magnitude: {a.magnitude}.")

print("Request its units in different ways (accessor methods):")
print(f"Units: {a.u}.")
print(f"Also units: {a.units}.")
```

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```
# Quantities can be compared between different units
# You can also build Quantity objects as follows:
a2 = 10 * UNITS.mm
print(f"Compare quantities built differently: {a == a2}")

# Quantities can be compared between different units
a2_diff_units = 1 * UNITS.cm
print(f"Compare quantities with different units: {a == a2_diff_units}")
```

```
Object a is a pint.Quantity: 10 millimeter
Request its magnitude in different ways (accessor methods):
Magnitude: 10.
Also magnitude: 10.
Request its units in different ways (accessor methods):
Units: millimeter.
Also units: millimeter.
Compare quantities built differently: True
Compare quantities with different units: True
```

PyAnsys Geometry objects work by returning `Quantity` objects whenever the property requested has a physical meaning.

Return `Quantity` objects for `Point3D` objects.

```
[3]: from ansys.geometry.core.math import Point3D
```

```
point_a = Point3D([1,2,4])
print("==== Point3D([1,2,4]) =====")
print(f"Point3D is a numpy.ndarray in SI units: {point_a}.")
print(f"However, request each of the coordinates individually...\n")
print(f"X Coordinate: {point_a.x}")
print(f"Y Coordinate: {point_a.y}")
print(f"Z Coordinate: {point_a.z}\n")

# Now, store the information with different units...
point_a_km = Point3D([1,2,4], unit=UNITS.km)
print("==== Point3D([1,2,4], unit=UNITS.km) =====")
print(f"Point3D is a numpy.ndarray in SI units: {point_a_km}.")
print(f"However, request each of the coordinates individually...\n")
print(f"X Coordinate: {point_a_km.x}")
print(f"Y Coordinate: {point_a_km.y}")
print(f"Z Coordinate: {point_a_km.z}\n")

# These points, although they are in different units, can be added together.
res = point_a + point_a_km

print("==== res = point_a + point_a_km =====")
print(f"numpy.ndarray: {res}")
print(f"X Coordinate: {res.x}")
print(f"Y Coordinate: {res.y}")
print(f"Z Coordinate: {res.z}")

==== Point3D([1,2,4]) =====
Point3D is a numpy.ndarray in SI units: [1. 2. 4.]
```

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```

However, request each of the coordinates individually...

X Coordinate: 1 meter
Y Coordinate: 2 meter
Z Coordinate: 4 meter

===== Point3D([1,2,4], unit=UNITS.km) =====
Point3D is a numpy.ndarray in SI units: [1000. 2000. 4000.].
However, request each of the coordinates individually...

X Coordinate: 1 kilometer
Y Coordinate: 2 kilometer
Z Coordinate: 4 kilometer

===== res = point_a + point_a_km =====
numpy.ndarray: [1001. 2002. 4004.]
X Coordinate: 1001.0 meter
Y Coordinate: 2002.0 meter
Z Coordinate: 4004.0 meter

```

Use default units

PyAnsys Geometry implements the concept of *default units*.

```

[4]: from ansys.geometry.core.misc import DEFAULT_UNITS

print("=== Default unit length ===")
print(DEFAULT_UNITS.LENGTH)

print("=== Default unit angle ===")
print(DEFAULT_UNITS.ANGLE)

=== Default unit length ===
meter
=== Default unit angle ===
radian

```

It is important to differentiate between *client-side* default units and *server-side* default units. You are able to control both of them.

Print the default server unit length.

```

[5]: print("=== Default server unit length ===")
print(DEFAULT_UNITS.SERVER_LENGTH)

=== Default server unit length ===
meter

```

Use default units.

```

[6]: from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import DEFAULT_UNITS

DEFAULT_UNITS.LENGTH = UNITS.mm

```

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```

point_2d_default_units = Point2D([3, 4])
print("This is a Point2D with default units")
print(f"X Coordinate: {point_2d_default_units.x}")
print(f"Y Coordinate: {point_2d_default_units.y}")
print(f"numpy.ndarray value: {point_2d_default_units}")

# Revert back to original default units
DEFAULT_UNITS.LENGTH = UNITS.m

```

```

This is a Point2D with default units
X Coordinate: 3 millimeter
Y Coordinate: 4 millimeter
numpy.ndarray value: [0.003 0.004]

```

PyAnsys Geometry has certain auxiliary classes implemented that provide proper unit checking when assigning values. Although they are basically intended for internal use of the library, you can define them for use.

```
[7]: from ansys.geometry.core.misc import Angle, Distance
```

Start with Distance. The main difference between a Quantity object (that is, from `pint import Quantity`) and a Distance is that there is an active check on the units passed (in case they are not the default ones). Here are some examples.

```
[8]: radius = Distance(4)
print(f"The radius is {radius.value}.")

# Reassign the value of the distance
radius.value = 7 * UNITS.cm
print(f"After reassignment, the radius is {radius.value}.")

# Change the units if desired
radius.unit = UNITS.cm
print(f"After changing its units, the radius is {radius.value}.")

```

```

The radius is 4 meter.
After reassignment, the radius is 0.07 meter.
After changing its units, the radius is 7.0000000000000001 centimeter.

```

The next two code examples show how unreasonable operations raise errors.

```
[9]: try:
    radius.value = 3 * UNITS.degrees
except TypeError as err:
    print(f"Error raised: {err}")

```

```
Error raised: The pint.Unit provided as an input should be a [length] quantity.
```

```
[10]: try:
    radius.unit = UNITS.fahrenheit
except TypeError as err:
    print(f"Error raised: {err}")

```

Error raised: The pint.Unit provided as an input should be a [length] quantity.

The same behavior applies to the Angle object. Here are some examples.

```
[11]: import numpy as np

rotation_angle = Angle(np.pi / 2)
print(f"The rotation angle is {rotation_angle.value}.")

# Try reassigning the value of the distance
rotation_angle.value = 7 * UNITS.degrees
print(f"After reassignment, the rotation angle is {rotation_angle.value}.")

# You could also change its units if desired
rotation_angle.unit = UNITS.degrees
print(f"After changing its units, the rotation angle is {rotation_angle.value}.")
```

The rotation angle is 1.5707963267948966 radian.
 After reassignment, the rotation angle is 0.12217304763960307 radian.
 After changing its units, the rotation angle is 7.0 degree.

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8.1.3 PyAnsys Geometry 101: Sketching

With PyAnsys Geometry, you can build powerful dynamic sketches without communicating with the Geometry service. This example shows how to build some simple sketches.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core.math import Plane, Point2D, Point3D, Vector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Add a box to sketch

The Sketch object is the starting point. Once it is created, you can dynamically add various curves to the sketch. Here are some of the curves that are available:

- arc
- box
- circle
- ellipse
- gear
- polygon
- segment
- slot
- trapezoid
- triangle

Add a box to the sketch.

```
[2]: sketch = Sketch()

sketch.segment(Point2D([0,0]), Point2D([0,1]))
sketch.segment(Point2D([0,1]), Point2D([1,1]))
sketch.segment(Point2D([1,1]), Point2D([1,0]))
sketch.segment(Point2D([1,0]), Point2D([0,0]))
```

```
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

A *functional-style sketching API* is also implemented. It allows you to append curves to the sketch with the idea of *never picking up your pen*.

Use the functional-style sketching API to add a box.

```
[3]: sketch = Sketch()

(
  sketch.segment(Point2D([0,0]), Point2D([0,1]))
    .segment_to_point(Point2D([1,1]))
    .segment_to_point(Point2D([1,0]))
    .segment_to_point(Point2D([0,0]))
)
```

```
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

A Sketch object uses the XY plane by default. You can define your own custom plane using three parameters: `origin`, `direction_x`, and `direction_y`.

Add a box on a custom plane.

```
[4]: plane = Plane(origin=Point3D([0,0,0]), direction_x=Vector3D([1,2,-1]), direction_
↳y=Vector3D([1,0,1]))

sketch = Sketch(plane)

sketch.box(Point2D([0,0]), 1, 1)

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

Combine concepts to create powerful sketches

Combine these simple concepts to create powerful sketches.

```
[5]: # Complex Fluent API Sketch - PCB

sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↳"LowerEdge")
    .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↳mm), tag="SupportedCorner")
    .segment_to_point(Point2D([41.5, 5], unit=UNITS.mm))
    .arc_to_point(Point2D([43, 6.5], unit=UNITS.mm), Point2D([43, 5], unit=UNITS.mm),_
↳True)
    .segment_to_point(Point2D([55, 6.5], unit=UNITS.mm))
    .arc_to_point(Point2D([56.5, 8], unit=UNITS.mm), Point2D([55, 8], unit=UNITS.mm))
    .segment_to_point(Point2D([56.5, 35], unit=UNITS.mm))
    .arc_to_point(Point2D([55, 36.5], unit=UNITS.mm), Point2D([55, 35], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 36.5], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 0], unit=UNITS.mm))
    .circle(Point2D([4, 4], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor1")
    .circle(Point2D([51, 34.5], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor2")
)

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

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8.1.4 PyAnsys Geometry 101: Modeling

Once you understand PyAnsys Geometry's mathematical constructs, units, and sketching capabilities, you can dive into its modeling capabilities.

PyAnsys Geometry is a Python client that connects to a modeling service. Here are the modeling services that are available for connection:

- **DMS:** Windows-based modeling service that has been containerized to ease distribution, execution, and remotability operations.
- **Geometry service:** Linux-based approach of DMS that is currently under development.
- **Ansys Discovery and SpaceClaim:** PyAnsys Geometry is capable of connecting to a running session of Ansys Discovery or SpaceClaim. Although this is not the main use case for PyAnsys Geometry, a connection to one of these Ansys products is possible. Because these products have graphical user interfaces, performance is not as high with this option as with the previous options. However, connecting to a running instance of Discovery or SpaceClaim might be useful for some users.

Launch a modeling service

While the PyAnsys Geometry operations in earlier examples did not require communication with a modeling service, this example requires that a modeling service is available. All subsequent examples also require that a modeling service is available.

Launch a modeling service session.

```
[1]: from ansys.geometry.core import launch_modeler
```

```
# Start a modeler session
modeler = launch_modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x1aa68fb4890)
```

```
Ansys Geometry Modeler Client (0x1aa68e3e900)
Target:      localhost:700
Connection: Healthy
```

You can also launch your own services and connect to them. For information on connecting to an existing service, see the [Modeler API](#) documentation.

Here is how the class architecture is implemented:

- **Modeler:** Handler object for the active service session. This object allows you to connect to an existing service by passing in a host and a port. It also allows you to create **Design** objects, which is where the modeling takes place. For more information, see the [Modeler API](#) documentation.
- **Design:** Root object of your assembly (tree). While a **Design** object is also a **Component** object, it has enhanced capabilities, including creating named selections, adding materials, and handling beam profiles. For more information, see the [Design API](#) documentation.
- **Component:** One of the main objects for modeling purposes. **Component** objects allow you to create bodies, subcomponents, beams, design points, planar surfaces, and more. For more information, see the [Component API](#) documentation.

The following code examples show how you use these objects. More capabilities of these objects are shown in the specific example sections for sketching and modeling.

Create and plot a sketch

Create a Sketch object and plot it.

```
[2]: from ansys.geometry.core.sketch import Sketch
      from ansys.geometry.core.math import Point2D
      from ansys.geometry.core.misc import UNITS, Distance

outer_hole_radius = Distance(0.5, UNITS.m)

sketch = Sketch()
(
    sketch.segment(start=Point2D([-4, 5], unit=UNITS.m), end=Point2D([4, 5], unit=UNITS.
↪m))
    .segment_to_point(end=Point2D([4, -5], unit=UNITS.m))
    .segment_to_point(end=Point2D([-4, -5], unit=UNITS.m))
    .segment_to_point(end=Point2D([-4, 5], unit=UNITS.m))
    .box(
        center=Point2D([0, 0], unit=UNITS.m),
        width=Distance(3, UNITS.m),
        height=Distance(3, UNITS.m),
    )
    .circle(center=Point2D([3, 4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([-3, -4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([-3, 4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([3, -4], unit=UNITS.m), radius=outer_hole_radius)
)

# Plot the sketch
sketch.plot()

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↪http-equiv="Content-...
```

Perform some modeling operations

Now that the sketch is ready to be extruded, perform some modeling operations, including creating the design, creating the body directly on the design, and plotting the body.

```
[3]: # Start by creating the Design
      design = modeler.create_design("ModelingDemo")

# Create a body directly on the design by extruding the sketch
body = design.extrude_sketch(
    name="Design_Body", sketch=sketch, distance=Distance(80, unit=UNITS.cm)
)

# Plot the body
design.plot()

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↪http-equiv="Content-...
```


Perform some operations on the body

Perform some operations on the body.

```
[4]: # Request its faces, edges, volume...
faces = body.faces
edges = body.edges
volume = body.volume

print(f"This is body {body.name} with ID (server-side): {body.id}.")
print(f"This body has {len(faces)} faces and {len(edges)} edges.")
print(f"The body volume is {volume}.")
```

```
This is body Design_Body with ID (server-side): 0:22.
This body has 14 faces and 32 edges.
The body volume is 54.28672587712814 meter ** 3.
```

Other operations that can be performed include adding a midsurface offset and thickness (only for planar bodies), imprinting curves, assigning materials, copying, and translating.

Copy the body on a new subcomponent and translate it.

```
[5]: from ansys.geometry.core.math import UNITVECTOR3D_X

# Create a component
comp = design.add_component("Component")

# Copy the body that belongs to this new component
body_copy = body.copy(parent=comp, name="Design_Component_Body")

# Displace this new body by a certain distance (10m) in a certain direction (X-axis)
body_copy.translate(direction=UNITVECTOR3D_X, distance=Distance(10, unit=UNITS.m))

# Plot the result of the entire design
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Create and assign materials to the bodies that were created.

```
[6]: from pint import Quantity

from ansys.geometry.core.materials import Material, MaterialProperty,
↳MaterialPropertyType

# Define some general properties for the material.
density = Quantity(125, 10 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
poisson_ratio = Quantity(0.33, UNITS.dimensionless)
tensile_strength = Quantity(45) # WARNING: If no units are defined,
#it is assumed that the magnitude is in the units expected by the server.

# Once your material properties are defined, you can easily create a material.
material = Material(
    "steel",
    density,
```

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(continued from previous page)

```

[MaterialProperty(MaterialPropertyType.POISSON_RATIO, "PoissonRatio", poisson_
↪ratio)],
)

# If you forgot to add a property, or you want to overwrite its value, you can still
# add properties to your created material.
material.add_property(
    type=MaterialPropertyType.TENSILE_STRENGTH, name="TensileProp", quantity=tensile_
↪strength
)

# Once your material is properly defined, send it to the server.
# This material can then be reused by different objects
design.add_material(material)

# Assign your material to your existing bodies.
body.assign_material(material)
body_copy.assign_material(material)

```

Currently materials do not have any impact on the visualization when plotting is requested, although this could be a future feature. If the final assembly is open in Discovery or SpaceClaim, you can observe the changes.

Create a named selection

PyAnsys Geometry supports the creation of a named selection via the Design object.

Create a named selection with some of the faces of the previous body and the body itself.

```

[7]: # Create a named selection
faces = body.faces
ns = design.create_named_selection("MyNamedSelection", bodies=[body], faces=[faces[0], ↪
↪faces[-1]])
print(f"This is a named selection called {ns.name} with ID (server-side): {ns.id}.")

```

This is a named selection called MyNamedSelection with ID (server-side): 0:427.

Perform deletions

Deletion operations for bodies, named selections, and components are possible, always from the scope expected. For example, if you attempted to delete the original body from a component that has no ownership over it (such as your comp object), the deletion would fail. If you attempted to perform this deletion from the design object, the deletion would succeed.

The next two code examples show how deletion works.

```

[8]: # If you try to delete this body from an "unauthorized" component, the deletion is not ↪
↪allowed.
comp.delete_body(body)
print(f"Is the body alive? {body.is_alive}")

# If you request a plot of the entire design, you can still see it.
design.plot()

```

Is the body alive? True

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv="Content-...
```

```
[9]: # Because the body belongs to the ``design`` object and not the ``comp`` object,
# deleting it from ``design`` object works.
design.delete_body(body)
print(f"Is the body alive? {body.is_alive}")

# If you request a plot of the entire design, it is no longer visible.
design.plot()
```

```
Is the body alive? False
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv="Content-...
```

Export files

Once modeling operations are finalized, you can export files in different formats. For the formats supported by DMS, see the `DesignFileFormat` class in the `Design` module documentation.

Export files in SCDOCX and FMD formats.

```
[10]: import os
from pathlib import Path

from ansys.geometry.core.designer import DesignFileFormat

# Path to downloads directory
file_dir = Path(os.getcwd(), "downloads")
file_dir.mkdir(parents=True, exist_ok=True)

# Download the model in different formats
design.download(file_location=Path(file_dir, "ModelingDemo.scdocx"), ↪
↪format=DesignFileFormat.SCDOCX)
design.download(file_location=Path(file_dir, "ModelingDemo.fmd"), ↪
↪format=DesignFileFormat.FMD)
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[11]: modeler.close()
```

Note

If the server session already existed (that is, it was not launched by the current client session), you cannot use this method to close the server session. You must manually close the server session instead. This is a safeguard for user-spawned services.

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8.1.5 PyAnsys Geometry 101: Plotter

This example provides an overview of PyAnsys Geometry’s plotting capabilities, focusing on its plotter features. After reviewing the fundamental concepts of sketching and modeling in PyAnsys Geometry, it shows how to leverage these key plotting capabilities:

- **Multi-object plotting:** You can conveniently plot a list of elements, including objects created in both PyAnsys Geometry and PyVista libraries.
- **Interactive object selection:** You can interactively select PyAnsys Geometry objects within the scene. This enables efficient manipulation of these objects in subsequent scripting.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity
import pyvista as pv

from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import GeometryPlotter
from ansys.geometry.core.sketch import Sketch
```

Launch modeling service

Launch a modeling service session.

```
[2]: from ansys.geometry.core import launch_modeler

# Start a modeler session
modeler = launch_modeler()
print(modeler)

Ansys Geometry Modeler (0x2181bf89ca0)

Ansys Geometry Modeler Client (0x2181b4472f0)
Target:    localhost:700
Connection: Healthy
```

You can also launch your own services and connect to them. For information on connecting to an existing service, see the [Modeler API](#) documentation.

Instantiate design and initialize object list

Instantiate a new design to work on and initialize a list of objects for plotting.

```
[3]: # init modeler
design = modeler.create_design("Multiplot")

plot_list = []
```

You are now ready to create some objects and use the plotter capabilities.

Create a PyAnsys Geometry body cylinder

Use PyAnsys Geometry to create a body cylinder.

```
[4]: cylinder = Sketch()
cylinder.circle(Point2D([10, 10], UNITS.m), 1.0)
cylinder_body = design.extrude_sketch("JustACyl", cylinder, Quantity(10, UNITS.m))
plot_list.append(cylinder_body)
```

Create a PyAnsys Geometry arc sketch

Use PyAnsys Geometry to create an arc sketch.

```
[5]: sketch = Sketch()
sketch.arc(
    Point2D([20, 20], UNITS.m),
    Point2D([20, -20], UNITS.m),
    Point2D([10, 0], UNITS.m),
    tag="Arc",
)
plot_list.append(sketch)
```

Create a PyVista cylinder

Use PyVista to create a cylinder.

```
[6]: cyl = pv.Cylinder(radius=5, height=20, center=(-20, 10, 10))
plot_list.append(cyl)
```

Create a PyVista multiblock

Use PyVista to create a multiblock with a sphere and a cube.

```
[7]: blocks = pv.MultiBlock(
    [pv.Sphere(center=(20, 10, -10), radius=10), pv.Cube(x_length=10, y_length=10, z_
    ↪length=10)]
)
plot_list.append(blocks)
```

Create a PyAnsys Geometry body box

Use PyAnsys Geometry to create a body box that is a cube.

```
[8]: box2 = Sketch()
      box2.box(Point2D([-10, 20], UNITS.m), Quantity(10, UNITS.m), Quantity(10, UNITS.m))
      box_body2 = design.extrude_sketch("JustABox", box2, Quantity(10, UNITS.m))
      plot_list.append(box_body2)
```

Plot objects

When plotting the created objects, you have several options.

You can simply plot one of the created objects.

```
[9]: plotter = GeometryPlotter()
      plotter.show(box_body2)
```

```
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↪http-equiv=&quot;Content-...
```

You can plot the whole list of objects.

```
[10]: plotter = GeometryPlotter()
       plotter.show(plot_list)
```

```
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↪http-equiv=&quot;Content-...
```

The Python visualizer is used by default. However, you can also use `trame` for visualization.

```
plotter = GeometryPlotter(use_trame=True)
plotter.show(plot_list)
```

Clip objects

You can clip any object represented in the plotter by defining a Plane object that intersects the target object.

```
[11]: from ansys.geometry.core.math import Plane, Point3D
      pl = GeometryPlotter()

      # Define PyAnsys Geometry box
      box2 = Sketch()
      box2.box(Point2D([-10, 20], UNITS.m), Quantity(10, UNITS.m), Quantity(10, UNITS.m))
      box_body2 = design.extrude_sketch("JustABox", box2, Quantity(10, UNITS.m))

      # Define plane to clip the box
      origin = Point3D([-10., 20., 5.], UNITS.m)
      plane = Plane(origin=origin, direction_x=[1, 1, 1], direction_y=[-1, 0, 1])

      # Add the object with the clipping plane
      pl.plot(box_body2, clipping_plane=plane)
      pl.show()
```

```
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↪http-equiv=&quot;Content-...
```

Select objects interactively

PyAnsys Geometry's plotter supports interactive object selection within the scene. This enables you to pick objects for subsequent script manipulation.

```
[12]: plotter = GeometryPlotter(allow_picking=True)

# Plotter returns picked bodies
picked_list = plotter.show(plot_list)
print(picked_list)

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↪http-equiv="Content-...
```

None

It is also possible to enable picking directly for a specific design or component object alone. In the following cell, picking is enabled for the design object.

```
[13]: picked_list = design.plot(allow_picking=True)
print(picked_list)

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↪http-equiv="Content-...
```

None

Render in different colors

You can render the objects in different colors automatically using PyVista's default color cyler. In order to do this, activate the `multi_colors=True` option when calling the `plot()` method.

In the following cell you can create a new design and plot a prism and a cylinder in different colors.

```
[14]: design = modeler.create_design("MultiColors")

# Create a sketch of a box
sketch_box = Sketch().box(Point2D([0, 0], unit=UNITS.m), width=30 * UNITS.m, height=40 *
↪UNITS.m)

# Create a sketch of a circle (overlapping the box slightly)
sketch_circle = Sketch().circle(Point2D([20, 0], unit=UNITS.m), radius=3 * UNITS.m)

# Extrude both sketches to get a prism and a cylinder
design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

# Design plotting
design.plot(multi_colors=True)

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↪http-equiv="Content-...
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[15]: modeler.close()
```

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8.2 Sketching examples

These examples demonstrate math operations on geometric objects and sketching capabilities, combined with server-based operations.

Download this example

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8.2.1 Sketching: Basic usage

This example shows how to use basic PyAnsys Geometry sketching capabilities.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core.misc.units import UNITS as u
      from ansys.geometry.core.sketch import Sketch
```

Create a sketch

Sketches are fundamental objects for drawing basic shapes like lines, segments, circles, ellipses, arcs, and polygons.

You create a `Sketch` instance by defining a drawing plane. To define a plane, you declare a point and two fundamental orthogonal directions.

```
[2]: from ansys.geometry.core.math import Plane, Point2D, Point3D
```

Define a plane for creating a sketch.

```
[3]: # Define the origin point of the plane
      origin = Point3D([1, 1, 1])

      # Create a plane located in previous point with desired fundamental directions
      plane = Plane(
```

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```

    origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
)

# Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

```

Draw shapes

To draw different shapes in the sketch, you use draw methods.

Draw a circle

You draw a circle in a sketch by specifying the center and radius.

```
[4]: sketch.circle(Point2D([2, 1]), radius=30 * u.cm, tag="Circle")
sketch.select("Circle")
sketch.plot_selection()
```

```
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↵http-equiv=&quot;Content-...
```

Draw an ellipse

You draw an ellipse in a sketch by specifying the center, major radius, and minor radius.

```
[5]: sketch.ellipse(
    Point2D([1, 1]), major_radius=2*u.m, minor_radius=1*u.m, tag="Ellipse"
)
sketch.select("Ellipse")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

Draw a polygon

You draw a regular polygon by specifying the center, radius, and desired number of sides.

```
[6]: sketch.polygon(
    Point2D([1, 1]), inner_radius=3*u.m, sides=5, tag="Polygon"
)
sketch.select("Polygon")
sketch.plot_selection()
```

```
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↵http-equiv=&quot;Content-...
```

Draw an arc

You draw an arc of circumference by specifying the center, starting point, and ending point.

```
[7]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
sketch.arc(start_point, end_point, Point2D([1,1]), tag="Arc")
sketch.select("Arc")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

There are also additional ways to draw arcs, such as by specifying the start, center point, and angle.

```
[8]: start_point = Point2D([2, 1], unit=u.m)
center_point = Point2D([1, 1], unit=u.m)
angle = 90
sketch.arc_from_start_center_and_angle(
    start_point, center_point, angle=90, tag="Arc_from_start_center_angle"
)
sketch.select("Arc_from_start_center_angle")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Or by specifying the start, end point, and radius.

```
[9]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
radius = 1 * u.m
sketch.arc_from_start_end_and_radius(
    start_point, end_point, radius, tag="Arc_from_start_end_radius"
)
sketch.select("Arc_from_start_end_radius")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Draw a slot

You draw a slot by specifying the center, width, and height.

```
[10]: sketch.slot(Point2D([2, 0]), 4, 3, tag="Slot")
sketch.select("Slot")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Draw a box

You draw a box by specifying the center, width, and height.

```
[11]: sketch.box(Point2D([2, 0]), 4, 5, tag="Box")
sketch.select("Box")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

Draw a segment

You draw a segment by specifying the starting point and ending point.

```
[12]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
sketch.segment(start_point, end_point, "Segment")
sketch.select("Segment")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

Plot the sketch

The `Plotter` class provides capabilities for plotting different PyAnsys Geometry objects. PyAnsys Geometry uses PyVista as the visualization backend.

You use the `plot_sketch` method to plot a sketch. This method accepts a `Sketch` instance and some extra arguments to further customize the visualization of the sketch. These arguments include showing the plane of the sketch and its frame.

```
[13]: # Plot the sketch in the whole scene
from ansys.geometry.core.plotting import GeometryPlotter

pl = GeometryPlotter()
pl.add_sketch(sketch, show_plane=True, show_frame=True)
pl.show()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

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8.2.2 Sketching: Dynamic sketch plane

The sketch is a lightweight, two-dimensional modeler driven primarily by client-side execution.

At any point, the current state of a sketch can be used for operations such as extruding a body, projecting a profile, or imprinting curves.

The sketch is designed as an effective *functional-style* API with all operations receiving 2D configurations.

For easy reuse of sketches across different regions of your design, you can move a sketch around the global coordinate system by modifying the plane defining the current sketch location.

This example creates a multi-layer PCB from many extrusions of the same sketch, creating unique design bodies for each layer.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_Z, Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile

You can create, modify, and plot Sketch instances independent of supporting Geometry service instances.

To define the sketch profile for the PCB, you create a sketch outline of individual Segment and Arc objects with two circular through-hole attachment points added within the profile boundary to maintain a single, closed sketch face.

Create a single Sketch instance to use for multiple design operations.

```
[2]: sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↳ "LowerEdge")
    .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↳ mm), tag="SupportedCorner")
    .segment_to_point(Point2D([41.5, 5], unit=UNITS.mm))
    .arc_to_point(Point2D([43, 6.5], unit=UNITS.mm), Point2D([43, 5], unit=UNITS.mm),
↳ True)
    .segment_to_point(Point2D([55, 6.5], unit=UNITS.mm))
    .arc_to_point(Point2D([56.5, 8], unit=UNITS.mm), Point2D([55, 8], unit=UNITS.mm))
    .segment_to_point(Point2D([56.5, 35], unit=UNITS.mm))
    .arc_to_point(Point2D([55, 36.5], unit=UNITS.mm), Point2D([55, 35], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 36.5], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 0], unit=UNITS.mm))
    .circle(Point2D([4, 4], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor1")
    .circle(Point2D([51, 34.5], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor2")
)

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
↳ http-equiv="Content-...
```

Instantiate the modeler

Launch a modeling service and connect to it.

```
[3]: modeler = launch_modeler()
print(modeler)

Ansys Geometry Modeler (0x1ae55a304a0)

Ansys Geometry Modeler Client (0x1ae55a329f0)
  Target:    localhost:7000
  Connection: Healthy
```

Extrude multiple bodies

Use the single sketch profile to extrude the board profile at multiple Z-offsets. Create a named selection from the resulting list of layer bodies.

Note that translating the sketch plane prior to extrusion is more effective (10 server calls) than creating a design body on the supporting server and then translating the body on the server (20 server calls).

```
[4]: design = modeler.create_design("ExtrudedBoardProfile")

layers = []
layer_thickness = Quantity(0.20, UNITS.mm)
for layer_index in range(10):
    layers.append(design.extrude_sketch(f"BoardLayer_{layer_index}", sketch, layer_
    ↪thickness))
    sketch.translate_sketch_plane_by_distance(UNITVECTOR3D_Z, layer_thickness)

board_named_selection = design.create_named_selection("FullBoard", bodies=layers)
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
    ↪http-equiv=&quot;Content-...
```

Close the modeler

Close the modeler to release the resources.

```
[5]: modeler.close()
```

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8.2.3 Sketching: Parametric sketching for gears

This example shows how to use gear sketching shapes from PyAnsys Geometry.

Perform required imports and pre-sketching operations

Perform required imports and instantiate the `Modeler` instance and the basic elements that define a sketch.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point2D, Point3D
from ansys.geometry.core.misc import UNITS, Distance
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.plotting import GeometryPlotter

# Start a modeler session
modeler = launch_modeler()
print(modeler)

# Define the origin point of the plane
origin = Point3D([1, 1, 1])

# Create a plane containing the previous point with desired fundamental directions
plane = Plane(
    origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
)

Ansys Geometry Modeler (0x24ff29bb860)

Ansys Geometry Modeler Client (0x24ff2ed8c20)
Target: localhost:700
Connection: Healthy
```

Sketch a dummy gear

`DummyGear` sketches are simple gears that have straight teeth. While they do not ensure actual physical functionality, they might be useful for some simple playground tests.

Instantiate a new `Sketch` object and then define and plot a dummy gear.

```
[2]: # Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

# Define dummy gear
#
origin = Point2D([0, 1], unit=UNITS.meter)
outer_radius = Distance(4, unit=UNITS.meter)
inner_radius = Distance(3.8, unit=UNITS.meter)
n_teeth = 30
sketch.dummy_gear(origin, outer_radius, inner_radius, n_teeth)

# Plot dummy gear
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

After creating the sketch, extrudes it.

```
[3]: # Create a design
design = modeler.create_design("AdvancedFeatures_DummyGear")

# Extrude your sketch
dummy_gear = design.extrude_sketch("DummyGear", sketch, Distance(1000, UNITS.mm))

# Plot the design
design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

Sketch a spur gear

SpurGear sketches are parametric CAD spur gears based on four parameters:

- **origin**: Center point location for the desired spur gear. The value must be a `Point2D` object.
- **module**: Ratio between the pitch circle diameter in millimeters and the number of teeth. This is a common parameter for spur gears. The value should be an integer or a float.
- **pressure_angle**: Pressure angle expected for the teeth of the spur gear. This is also a common parameter for spur gears. The value must be a `pint.Quantity` object.
- **n_teeth**: Number of teeth. The value must be an integer.

Instantiate a new `Sketch` object and then define and plot a spur gear.

```
[4]: # Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

# Define spur gear
#
origin = Point2D([0, 1], unit=UNITS.meter)
module = 40
pressure_angle = Quantity(20, UNITS.deg)
n_teeth = 22

# Sketch spur gear
sketch.spur_gear(origin, module, pressure_angle, n_teeth)

# Plot spur gear
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

After creating the sketch, extrude it.

```
[5]: # Create a design
design = modeler.create_design("AdvancedFeatures_SpurGear")
```

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```
# Extrude sketch
dummy_gear = design.extrude_sketch("SpurGear", sketch, Distance(200, UNITS.mm))

# Plot design
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

Close the modeler

```
[6]: modeler.close()
```

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8.3 Modeling examples

These examples demonstrate service-based modeling operations.

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8.3.1 Modeling: Single body with material assignment

In PyAnsys Geometry, a *body* represents solids or surfaces organized within the `Design` assembly. The current state of sketch, which is a client-side execution, can be used for the operations of the geometric design assembly.

The Geometry service provides data structures to create individual materials and their properties. These data structures are exposed through PyAnsys Geometry.

This example shows how to create a single body from a sketch by requesting its extrusion. It then shows how to assign a material to this body.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.materials import Material, MaterialProperty,
↳MaterialPropertyType
from ansys.geometry.core.math import UNITVECTOR3D_Z, Frame, Plane, Point2D, Point3D,
↳
```

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```
↪UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Create sketch

Create a Sketch instance and insert a circle with a radius of 10 millimeters in the default plane.

```
[2]: sketch = Sketch()
      sketch.circle(Point2D([10, 10], UNITS.mm), Quantity(10, UNITS.mm))

[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x23d485bfbf0>
```

Initiate design on server

Launch a modeling service session and initiate a design on the server.

```
[3]: # Start a modeler session
      modeler = launch_modeler()
      print(modeler)

      design_name = "ExtrudeProfile"
      design = modeler.create_design(design_name)

Ansys Geometry Modeler (0x23d487a1e80)

Ansys Geometry Modeler Client (0x23d0637c260)
  Target:    localhost:700
  Connection: Healthy
```

Add materials to design

Add materials and their properties to the design. Material properties can be added when creating the Material object or after its creation. This code adds material properties after creating the Material object.

```
[4]: density = Quantity(125, 10 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
      poisson_ratio = Quantity(0.33, UNITS.dimensionless)
      tensile_strength = Quantity(45)
      material = Material(
          "steel",
          density,
          [MaterialProperty(MaterialPropertyType.POISSON_RATIO, "PoissonRatio", poisson_
↪ratio)],
      )
      material.add_property(MaterialPropertyType.TENSILE_STRENGTH, "TensileProp", Quantity(45))
      design.add_material(material)
```

Extrude sketch to create body

Extrude the sketch to create the body and then assign a material to it.

```
[5]: # Extrude the sketch to create the body
      body = design.extrude_sketch("SingleBody", sketch, Quantity(10, UNITS.mm))
```

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```
# Assign a material to the body
body.assign_material(material)

body.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv="Content-...
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[6]: modeler.close()
```

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8.3.2 Modeling: Rectangular plate with multiple bodies

You can create multiple bodies from a single sketch by extruding the same sketch in different planes.

The sketch is designed as an effective *functional-style* API with all operations receiving 2D configurations. For more information, see the *Sketch* user guide.

In this example, a box is located in the center of the plate, with the default origin of a sketch plane (origin at (0, 0, 0)). Four holes of equal radius are sketched at the corners of the plate. The plate is then extruded, leading to the generation of the requested body. The projection is at the center of the face. The default projection depth is through the entire part.

Perform required imports

Perform the required imports.

```
[1]: import numpy as np
from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point3D, Point2D, UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile

The sketch profile for the proposed design requires four segments that constitute the outer limits of the design, a box on the center, and a circle at its four corners.

You can use a single sketch instance for multiple design operations, including extruding a body, projecting a profile, and imprinting curves.

Define the sketch profile for the rectangular plate with multiple bodies.

```
[2]: sketch = Sketch()
(sketch.segment(Point2D([-4, 5], unit=UNITS.m), Point2D([4, 5], unit=UNITS.m))
 .segment_to_point(Point2D([4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, 5], unit=UNITS.m))
 .box(Point2D([0,0], unit=UNITS.m), Quantity(3, UNITS.m), Quantity(3, UNITS.m))
 .circle(Point2D([3, 4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 .circle(Point2D([-3, -4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 .circle(Point2D([-3, 4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 .circle(Point2D([3, -4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 )
```

```
[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x2d777b61b80>
```

Extrude sketch to create design

Establish a server connection and use the single sketch profile to extrude the base component at the Z axis. Create a named selection from the resulting list of bodies. In only three server calls, the design extrudes the four segments with the desired thickness.

```
[3]: modeler = launch_modeler()

design = modeler.create_design("ExtrudedPlate")

body = design.extrude_sketch(f"PlateLayer", sketch, Quantity(2, UNITS.m))

board_named_selection = design.create_named_selection("Plate", bodies=[body])
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

Add component with a planar surface

After creating a plate as a base component, you might want to add a component with a planar surface to it.

Create a sketch instance and then create a surface in the design with this sketch. For the sketch, it creates an ellipse, keeping the origin of the plane as its center.

```
[4]: # Add components to the design
planar_component = design.add_component("PlanarComponent")

# Initiate ``Sketch`` to create the planar surface.
planar_sketch = Sketch()
planar_sketch.ellipse(
    Point2D([0, 0], UNITS.m), Quantity(1, UNITS.m), Quantity(0.5, UNITS.m)
```

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```

)

planar_body = planar_component.create_surface("PlanarComponentSurface", planar_sketch)

comp_str = repr(planar_component)
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv="Content-...
```

Extrude from face to create body

Extrude a face profile by a given distance to create a solid body. There are no modifications against the body containing the source face.

```
[5]: longer_body = design.extrude_face(
    "LongerEllipseFace", planar_body.faces[0], Quantity(5, UNITS.m)
)
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv="Content-...
```

Translate body within plane

Use the `:func:translate()` `<ansys.geometry.core.designer.body.Body.translate>` method to move the body in a specified direction by a given distance. You can also move a sketch around the global coordinate system. For more information, see the *Dynamic Sketch Plane* example.

```
[6]: longer_body.translate(UnitVector3D([1, 0, 0]), Quantity(4, UNITS.m))
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv="Content-...
```

Close the modeler

Close the modeler to free up resources and release the connection.

```
[7]: modeler.close()
```

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8.3.3 Modeling: Extruded plate with cut operations

As seen in the *Rectangular plate with multiple bodies* example, you can create a complex sketch with holes and extrude it to create a body. However, you can also perform cut operations on the extruded body to achieve similar results.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point3D, Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile without holes

Create a sketch profile for the proposed design. The sketch is the same as the *Rectangular plate with multiple bodies* example, but without the holes.

These holes are created by performing cut operations on the extruded body in the next steps.

```
[2]: sketch = Sketch()
(sketch.segment(Point2D([-4, 5], unit=UNITS.m), Point2D([4, 5], unit=UNITS.m))
 .segment_to_point(Point2D([4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, 5], unit=UNITS.m))
 .box(Point2D([0,0], unit=UNITS.m), Quantity(3, UNITS.m), Quantity(3, UNITS.m))
)

modeler = launch_modeler()
design = modeler.create_design("ExtrudedPlateNoHoles")
body = design.extrude_sketch(f"PlateLayer", sketch, Quantity(2, UNITS.m))

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Define sketch profile for holes

Create a sketch profile for the holes in the proposed design. The holes are created by sketching circles at the four corners of the plate. First create a reference sketch for all the circles. This sketch is translated to the four corners of the plate.

```
[3]: sketch_hole = Sketch()
sketch_hole.circle(Point2D([0, 0], unit=UNITS.m), Quantity(0.5, UNITS.m))

hole_centers = [
    Plane(Point3D([3, 4, 0], unit=UNITS.m)),
    Plane(Point3D([-3, 4, 0], unit=UNITS.m)),
    Plane(Point3D([-3, -4, 0], unit=UNITS.m)),
    Plane(Point3D([3, -4, 0], unit=UNITS.m)),
]
```

Perform cut operations on the extruded body

Perform cut operations on the extruded body to create holes at the four corners of the plate.

```
[4]: for center in hole_centers:
      sketch_hole.plane = center
      design.extrude_sketch(
          name= f"H_{center.origin.x}_{center.origin.y}",
          sketch=sketch_hole,
          distance=Quantity(2, UNITS.m),
          cut=True,
      )

design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
<http-equiv=&quot;Content-...
```

Close the modeler

Close the modeler to free up resources and release the connection.

```
[5]: modeler.close()
```

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8.3.4 Modeling: Tessellation of two bodies

This example shows how to create two stacked bodies and return the tessellation as two merged bodies.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D, Point3D, Plane
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Create design

Create the basic sketches to be tessellated and extrude the sketch in the required plane. For more information on creating a component and extruding a sketch in the design, see the *Rectangular plate with multiple bodies* example.

Here is a typical situation in which two bodies, with different sketch planes, merge each body into a single dataset. This effectively combines all the faces of each individual body into a single dataset without separating faces.

```
[2]: modeler = launch_modeler()

sketch_1 = Sketch()
box = sketch_1.box(
    Point2D([10, 10], unit=UNITS.m), width=Quantity(10, UNITS.m), height=Quantity(5, UNITS.m)
)
circle = sketch_1.circle(
    Point2D([0, 0], unit=UNITS.m), radius=Quantity(25, UNITS.m)
)

design = modeler.create_design("TessellationDesign")
comp = design.add_component("TessellationComponent")
body = comp.extrude_sketch("Body", sketch=sketch_1, distance=10 * UNITS.m)

# Create the second body in a plane with a different origin
sketch_2 = Sketch(Plane([0, 0, 10]))
box = sketch_2.box(Point2D(
    [10, 10], unit=UNITS.m), width=Quantity(10, UNITS.m), height=Quantity(5, UNITS.m)
)
circle = sketch_2.circle(
    Point2D([0, 10], unit=UNITS.m), radius=Quantity(25, UNITS.m)
)

body = comp.extrude_sketch("Body", sketch=sketch_2, distance=10 * UNITS.m)
```

Tessellate component as two merged bodies

Tessellate the component and merge each body into a single dataset. This effectively combines all the faces of each individual body into a single dataset without separating faces.

```
[3]: dataset = comp.tessellate()
dataset
```

```
[3]: PolyData (0x132b221df60)
  N Cells:    3280
  N Points:   3300
  N Strips:    0
  X Bounds:   -2.500e+01, 2.500e+01
  Y Bounds:   -2.500e+01, 3.500e+01
  Z Bounds:   0.000e+00, 2.000e+01
  N Arrays:   0
```

Single body tessellation is possible. In that case, users can request the body-level tessellation method to tessellate the body and merge all the faces into a single dataset.

```
[4]: dataset = comp.bodies[0].tessellate()
dataset
```

```
[4]: MultiBlock (0x132b221dea0)
     N Blocks      7
     X Bounds     -25.000, 25.000
     Y Bounds     -24.999, 24.999
     Z Bounds      0.000, 10.000
```

Plot design

Plot the design.

```
[5]: design.plot()
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Close the modeler

Close the modeler.

```
[6]: modeler.close()
```

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8.3.5 Modeling: Design organization

The Design instance creates a design project within the remote Geometry service to complete all CAD modeling against.

You can organize all solid and surface bodies in each design within a customizable component hierarchy. A component is simply an organization mechanism.

The top-level design node and each child component node can have one or more bodies assigned and one or more components assigned.

The API requires each component of the design hierarchy to be given a user-defined name.

There are several design operations that result in a body being created within a design. Executing each of these methods against a specific component instance explicitly specifies the node of the design tree to place the new body under.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, Point2D
from ansys.geometry.core.misc import UNITS, Distance
from ansys.geometry.core.sketch import Sketch
```

Organize design

Extrude two sketches to create bodies. Assign the cylinder to the top-level design component. Assign the slot to the component nested one level beneath the top-level design component.

```
[2]: modeler = launch_modeler()

design = modeler.create_design("DesignHierarchyExample")

circle_sketch = Sketch()
circle_sketch.circle(Point2D([10, 10], UNITS.mm), Distance(10, UNITS.mm))

cylinder_body = design.extrude_sketch("10mmCylinder", circle_sketch, Distance(10, UNITS.
↳mm))

slot_sketch = Sketch()
slot_sketch.slot(Point2D([40, 10], UNITS.mm), Distance(20, UNITS.mm), Distance(10, UNITS.
↳mm))

nested_component = design.add_component("NestedComponent")
slot_body = nested_component.extrude_sketch("SlotExtrusion", slot_sketch, Distance(20,
↳UNITS.mm))

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
↳http-equiv=&quot;Content-...
```

Create nested component

Create a component that is nested under the previously created component and then create another cylinder from the previously used sketch.

```
[3]: double_nested_component = nested_component.add_component("DoubleNestedComponent")

circle_surface_body = double_nested_component.create_surface("CircularSurfaceBody",
↳circle_sketch)
circle_surface_body.translate(UNITVECTOR3D_X, Distance(-35, UNITS.mm))

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
↳http-equiv=&quot;Content-...
```

Use surfaces from body to create additional bodies

You can use surfaces from any body across the entire design as references for creating additional bodies.

Extrude a cylinder from the surface body assigned to the child component.

```
[4]: cylinder_from_face = nested_component.extrude_face("CylinderFromFace", circle_surface_
↳body.faces[0], Distance(30, UNITS.mm))
cylinder_from_face.translate(UNITVECTOR3D_X, Distance(-25, UNITS.mm))

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

Close the modeler

Close the modeler to release the resources.

```
[5]: modeler.close()
```

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8.3.6 Modeling: Boolean operations

This example shows how to use Boolean operations for geometry manipulation.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.designer import Body
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import GeometryPlotter
from ansys.geometry.core.sketch import Sketch
```

Launch local modeler

Launch the local modeler. If you are not familiar with how to launch the local modeler, see the “Launch a modeling service” section in the *PyAnsys Geometry 101: Modeling* example.

```
[2]: modeler = launch_modeler()
print(modeler)

Ansys Geometry Modeler (0x226a1770500)

Ansys Geometry Modeler Client (0x226a2004da0)
  Target:    localhost:700
  Connection: Healthy
```

Define bodies

This section defines the bodies to use the Boolean operations on. First you create sketches of a box and a circle, and then you extrude these sketches to create 3D objects.

Create sketches

Create sketches of a box and a circle that serve as the basis for your bodies.

```
[3]: # Create a sketch of a box
sketch_box = Sketch().box(Point2D([0, 0], unit=UNITS.m), width=30 * UNITS.m, height=40 *
↳UNITS.m)

# Create a sketch of a circle (overlapping the box slightly)
sketch_circle = Sketch().circle(Point2D([20, 0], unit=UNITS.m), radius=10 * UNITS.m)
```

Extrude sketches

After the sketches are created, extrude them to create 3D objects.

```
[4]: # Create a design
design = modeler.create_design("example_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

You must extrude the sketches each time that you perform an example operation. This is because performing a Boolean operation modifies the underlying design permanently. Thus, you no longer have two bodies. As shown in the Boolean operations themselves, whenever you pass in a body, it is consumed, and so it no longer exists. The remaining body (with the performed Boolean operation) is the one that performed the call to the method.

Select bodies

You can optionally select bodies in the plotter as described in the “Select objects interactively” section in the *PyAnsys Geometry 101: Plotter* example. As shown in this example, the plotter preserves the picking order, meaning that the output list is sorted according to the picking order.

```
pl = GeometryPlotter(allow_picking=True)
pl.plot(design.bodies)
pl.show()
bodies: list[Body] = GeometryPlotter(allow_picking=True).show(design.bodies)
```

Otherwise, you can select bodies from the design directly.

```
[5]: bodies = [design.bodies[0], design.bodies[1]]
```

Perform Boolean operations

This section performs Boolean operations on the defined bodies using the PyAnsys Geometry library. It explores intersection, union, and subtraction operations.

Perform an intersection operation

To perform an intersection operation on the bodies, first set up the bodies.

```
[6]: # Create a design
design = modeler.create_design("intersection_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

Perform the intersection and plot the results.

```
[7]: prism.intersect(cylin)
_ = GeometryPlotter().show(design.bodies)
```

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<http-equiv="Content-...

The final remaining body is the prism body because the cylin body has been consumed.

```
[8]: print(design.bodies)

[
ansys.geometry.core.designer.Body 0x226a24a9b20
  Name          : Prism
  Exists        : True
  Parent component : intersection_design
  MasterBody     : 0:22
  Surface body   : False
  Color          : #D6F7D1
]
```

Perform a union operation

To carry out a union operation on the bodies, first set up the bodies.

```
[9]: # Create a design
design = modeler.create_design("union_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

Perform the union and plot the results.

```
[10]: prism.unite(cylin)
_ = GeometryPlotter().show(design.bodies)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv="Content-...
```

The final remaining body is the prism body because the cylin body has been consumed.

```
[11]: print(design.bodies)
```

```
[
ansys.geometry.core.designer.Body 0x226a3b3e210
  Name          : Prism
  Exists        : True
  Parent component : union_design
  MasterBody    : 0:22
  Surface body  : False
  Color         : #D6F7D1
]
```

Perform a subtraction operation

To perform a subtraction operation on the bodies, first set up the bodies.

```
[12]: # Create a design
design = modeler.create_design("subtraction_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

Perform the subtraction and plot the results.

```
[13]: prism.subtract(cylin)
_ = GeometryPlotter().show(design.bodies)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv="Content-...
```

The final remaining body is the prism body because the cylin body has been consumed.

```
[14]: print(design.bodies)
```

```
[
ansys.geometry.core.designer.Body 0x226a40bae0
  Name          : Prism
  Exists        : True
  Parent component : subtraction_design
  MasterBody    : 0:22
  Surface body  : False
  Color         : #D6F7D1
]
```

If you perform this action inverting the order of the bodies (that is, `cylin.subtract(prism)`), you can see the difference in the resulting shape of the body.

```
[15]: # Create a design
design = modeler.create_design("subtraction_design_inverted")
```

(continues on next page)

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```

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

# Invert subtraction
cylin.subtract(prism)
_ = GeometryPlotter().show(design.bodies)

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↵http-equiv="Content-...

```

In this case, the final remaining body is the cylin body because the prism body has been consumed.

```

[16]: print(design.bodies)

[
ansys.geometry.core.designer.Body 0x226a414be90
  Name          : Cylinder
  Exists        : True
  Parent component : subtraction_design_inverted
  MasterBody     : 0:85
  Surface body   : False
  Color         : #D6F7D1
]

```

Close the modeler

Close the modeler to release the resources.

```
[17]: modeler.close()
```

Summary

These Boolean operations provide powerful tools for creating complex geometries and combining or modifying existing shapes in meaningful ways.

Feel free to experiment with different shapes, sizes, and arrangements to further enhance your understanding of Boolean operations in PyAnsys Geometry and their applications.

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8.3.7 Modeling: Scale, map and mirror bodies

The purpose of this notebook is to demonstrate the `map()` and `scale()` functions and their usage for transforming bodies.

```
[1]: # Imports
import numpy as np

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
    Frame,
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
Ansys Geometry Modeler (0x1aab01417f0)
```

```
Ansys Geometry Modeler Client (0x1aab0206810)
  Target:    localhost:700
  Connection: Healthy
```

Scale body

The `scale()` function is designed to modify the size of 3D bodies by a specified scale factor. This function is an important part of geometric transformations, allowing for the dynamic resizing of bodies.

Usage of `scale()`

To use the `scale()` function, you call it on an instance of a geometry body, passing a single argument: the scale value. This value is a real number (Real) that determines the factor by which the body's size is changed.

```
body.scale(value)
```

Example: Making a cube

The following code snippets show how to change the size of a cube using the `scale()` function in Body objects. The process involves initializing a sketch design for the cube, defining the shape parameters, and then performing a rescaling operation to generate the new shape.

Initialize the cube sketch design

A new design sketch named “cube” is created.

```
[3]: design = m.create_design("cube")
```

Define cube parameters

side_length is set to 10 units, representing the side length of the cube.

```
[4]: # Cube parameters
side_length = 10
```

Create the profile cube

A square box is created centered on the origin using side_length as the side length of the square.

```
[5]: # Square with side length 10
box_sketch = Sketch().box(Point2D([0, 0]), side_length, side_length)

box_sketch.plot()

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↪http-equiv=&quot;Content-...
```

Create cube body

extrude_sketch on the box_sketch as the base sketch and create the 3D cube with distance being the side_length.

```
[6]: # Extrude the cube profile by a distance of side_length
cube = design.extrude_sketch("box", box_sketch, side_length)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Making the cube twice as large

- Copy the original cube. Using scale() with a value of 2, double the side lengths of the cube, thereby making the body twice as large, and then offset it to view the difference.

```
[7]: # Copy the original cube
doubled = cube.copy(cube.parent_component, "doubled_box")
# Double the size
doubled.scale(2)
# Translate the copied cube in the x direction
doubled.translate(UNITVECTOR3D_X, 30)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```


Halving the size of the *original* cube

Copy the original cube. Using `scale()` with a value of 0.5 effectively halves the side lengths of the cube. Then, offset the new cube to view the difference.

Note: Because the size of the cube in the previous cell was doubled, using the 0.25 factor translates it to half the size of the original cube.

```
[8]: # Copy the original cube
halved = cube.copy(cube.parent_component, "halved_box")
# Half the size
halved.scale(0.5)
# Translate the copied cube in the x direction
halved.translate(UNITVECTOR3D_X, -25)

design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Map body

The `map()` function enables the reorientation of 3D bodies by mapping them onto a new specified frame. This function is used for adjusting the orientation of geometric bodies within 3D space to match specific reference frames. With this function, you are able to effectively perform translation and rotation operations in a single method by specifying a new frame.

Usage of `map()`

To use the `map()` function, invoke it on an instance of a geometry body with a single argument: the new frame to map the body to. The frame is a structure or object that defines the new orientation parameters for the body.

```
body.map(new_frame)
```

Example: Creating an asymmetric cube

The following code snippets show how to use the `map()` function to reframe a cube body in the `Body` object. The process involves initializing a sketch design for the custom body, extruding the profile by a distance, and then performing the mapping operation to rotate the shape.

Initialize the shape sketch design

A new design sketch named “asymmetric_cube” is created.

```
[9]: # Initialize the sketch design
design = m.create_design("asymmetric_cube")
```

Create an asymmetric sketch profile

Make a sketch profile that is basically a cube centered on the origin with a side length of 2 with a cutout.

```
[10]: # Create the cube profile with a cut through it
asymmetric_profile = Sketch()
(
```

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```

asymmetric_profile.segment(Point2D([1, 1]), Point2D([-1, 1]))
.asymmetric_profile.segment_to_point(Point2D([0, 0.5]))
.asymmetric_profile.segment_to_point(Point2D([-1, -1]))
.asymmetric_profile.segment_to_point(Point2D([1, -1]))
.asymmetric_profile.segment_to_point(Point2D([1, 1]))
)

asymmetric_profile.plot()
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...

```

Create the asymmetric body

extrude_sketch on the asymmetric_profile as the base sketch, creating the 3D cube with a cutout, with the distance being 1.

```

[11]: # Extrude the asymmetric profile by a distance of 1 unit
body = design.extrude_sketch("box", asymmetric_profile, 1)

design.plot()
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...

```

Apply map reframing

First make a copy of the shape and translate it in 3D space so that you can view them side by side. Then, apply the reframing to the copied shape.

Note: The following map uses the default x direction, but the y direction is swapped with the z direction, effectively rotating the original shape so that it is standing vertically.

```

[12]: # Copy the body
copied_body = body.copy(body.parent_component, "copied_body")
# Apply the reframing
copied_body.map(Frame(Point3D([0, 0, 0]), UNITVECTOR3D_X, UNITVECTOR3D_Z))
# Shift the new modified body in the plane in the negative y direction by 2 units
copied_body.translate(UNITVECTOR3D_Y, -2)

design.plot()
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...

```

Mirror body

The mirror() function is designed to mirror the geometry of a body across a specified plane. This function plays a role in geometric transformations, enabling the reflection of bodies to create symmetrical designs.

Usage of mirror()

To use the `mirror()` function, you call it on an instance of a geometry body, passing a single argument: the plane across which to mirror the body. This plane is represented by a `Plane` object, defining the axis of symmetry for the mirroring operation.

```
body.mirror(plane)
```

Example: Triangle body

The following code snippets show how to use the `mirror()` function to reframe a cube body in the `Body` object. The process involves initializing a sketch design for the body profile, extruding the profile by a distance, and then performing the mirroring operation to reflect the shape over the specified axis.

Initialize the shape sketch design

A new design sketch named “triangle” is created.

```
[13]: # Initialize the sketch design
design = m.create_design("triangle")
```

Define parameters

`point1`: First vertex of the triangle. `point2`: Second vertex of the triangle. `point3`: Third vertex of the triangle.

```
[14]: point1 = Point2D([5, 0])
point2 = Point2D([2.5, 2.5])
point3 = Point2D([2.5, -2.5])
```

Create triangle sketch profile

Using `point1`, `point2`, and `point3`, define the vertices of the triangle profile using those three points and then create line segments connecting them.

```
[15]: # Draw the triangle sketch profile
sketch = Sketch()
sketch.segment(start=point1, end=point2)
sketch.segment(start=point2, end=point3)
sketch.segment(start=point3, end=point1)

sketch.plot()

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↪http-equiv="Content-...
```

Create triangular body

Using the sketch profile created in the previous step, use the `extrude_sketch` method to create a solid body with a depth of 1.

```
[16]: # Extrude the triangular body by a distance of 1
triangle = design.extrude_sketch("triangle_body", sketch, 1)

design.plot()
```

```
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↵http-equiv="Content-...
```

Mirror the triangular body

First, make a copy of the triangular body. Then, using `mirror()`, you can mirror the copied body over the ZY plane.

```
[17]: # Copy triangular body
mirrored_triangle = triangle.copy(triangle.parent_component, "mirrored_triangle")
# Mirror the copied body over the ZY plane (specified by the (0, 1, 0) and
# (0, 0, 1) unit vectors)

mirrored_triangle.mirror(Plane(direction_x=UNITVECTOR3D_Y, direction_y=UNITVECTOR3D_Z))

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv="Content-...
```

Closing the modeler

```
[18]: m.close()
```

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8.3.8 Modeling: Sweep chain and sweep sketch

This example shows how use the `sweep_sketch()` and `sweep_chain()` functions to create more complex extrusion profiles. You use the `sweep_sketch()` function with a closed sketch profile and the `sweep_chain()` function for an open profile.

```
[1]: # Imports
import numpy as np

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
```

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```

UNITVECTOR3D_Y,
UNITVECTOR3D_Z,
)
from ansys.geometry.core.shapes import Circle, Ellipse, Interval
from ansys.geometry.core.sketch import Sketch

```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```

Ansys Geometry Modeler (0x18a66aaacf0)

```

```

Ansys Geometry Modeler Client (0x18a66aabb30)
  Target:    localhost:700
  Connection: Healthy

```

Example: Creating a donut

The following code snippets show how to use the `sweep_sketch()` function to create a 3D donut shape in the `Design` object. The process involves initializing a sketch design for the donut, defining two circles for the profile and path, and then performing a sweep operation to generate the donut shape.

Initialize the donut sketch design

A new design sketch named “donut” is created.

```
[3]: # Initialize the donut sketch design
design_sketch = m.create_design("donut")
```

Define circle parameters

`path_radius` is set to 5 units, representing the radius of the circular path that the profile circle sweeps along. `profile_radius` is set to 2 units, representing the radius of the profile circle that sweeps along the path to create the donut body.

```
[4]: # Donut parameters
path_radius = 5
profile_radius = 2
```

Create the profile circle

A circle is created on the XZ-plane centered at the coordinates (5, 0, 0) with a radius defined by `profile_radius`. This circle serves as the profile or cross-sectional shape of the donut.

```
[5]: # Create the circular profile on the XZ plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(direction_x=UNITVECTOR3D_X, direction_y=UNITVECTOR3D_Z)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([path_radius, 0]), profile_radius)
```

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```
profile.plot()
```

```
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↵http-equiv=&quot;Content-...
```

Create the path circle

Another circle, representing the path along which the profile circle is swept, is created on the XY-plane centered at (0, 0, 0). The radius of this circle is defined by `path_radius`.

```
[6]: # Create the circular path on the XY plane centered at (0, 0, 0) with radius 5
path = [Circle(Point3D([0, 0, 0]), path_radius).trim(Interval(0, 2 * np.pi))]
```

Perform the sweep operation

The sweep operation uses the profile circle and sweeps it along the path circle to create the 3D body of the donut. The result is stored in the variable `body`.

```
[7]: # Perform the sweep and examine the final body created
body = design_sketch.sweep_sketch("donutsweep", profile, path)
```

```
design_sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

Example: Creating a bowl

This code demonstrates the process of using the `sweep_chain()` function to create a 3D model of a stretched bowl in the Design object. The model is generated by defining a quarter-ellipse as a profile and sweeping it along a circular path, creating a bowl shape with a stretched profile.

Initialize the bowl design

A design chain named “bowl” is created to initiate the bowl design process.

```
[8]: # Initialize the bowl sketch design
design_chain = m.create_design("bowl")
```

Define parameters

`radius` is set to 10 units, used for both the profile and path definitions.

```
[9]: # Define the radius parameter
radius = 10
```

Create the profile shape

A quarter-ellipse profile is created with a major radius equal to 10 units and a minor radius equal to 5 units. The ellipse is defined in a 3D space with a specific orientation and then trimmed to a quarter using an interval from 0 to $\pi/2$ radians. This profile shapes the bowl’s side.

```
[10]: # Create quarter-ellipse profile with major radius = 10, minor radius = 5
profile = [
    Ellipse(
        Point3D([0, 0, radius / 2]),
        radius,
        radius / 2,
        reference=UNITVECTOR3D_X,
        axis=UNITVECTOR3D_Y,
    ).trim(Interval(0, np.pi / 2))
]
```

Create the path

A circular path is created, positioned parallel to the XY-plane but shifted upwards by 5 units (half the major radius). The circle has a radius of 10 units and is trimmed to form a complete loop with an interval from 0 to 2pi radians. This path defines the sweeping trajectory for the profile to create the bowl.

```
[11]: # Create circle on the plane parallel to the XY-plane but moved up
# by 5 units with radius 10
path = [Circle(Point3D([0, 0, radius / 2]), radius).trim(Interval(0, 2 * np.pi))]
```

Perform the sweep operation

The bowl body is generated by sweeping the quarter-ellipse profile along the circular path. The sweep operation molds the profile shape along the path to form the stretched bowl. The result of this operation is stored in the variable body.

```
[12]: # Create the bowl body
body = design_chain.sweep_chain("bowl_sweep", path, profile)

design_chain.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

Closing the modeler

```
[13]: # Close the modeler
m.close()
```

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8.3.9 Modeling: Revolving a sketch

This example shows how to use the `revolve_sketch()` method to revolve a sketch around an axis to create a 3D body. You can also specify the angle of revolution to create a partial body.

```
[1]: # Imports
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.misc import UNITS, Angle
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
Ansys Geometry Modeler (0x17e8eb55850)
```

```
Ansys Geometry Modeler Client (0x17e8efe38c0)
  Target:    localhost:700
  Connection: Healthy
```

Example: Creating a quarter of a donut

The following code snippets show how to use the `revolve_sketch()` function to create a quarter of a 3D donut. The process involves defining a quarter of a circle as a profile and then revolving it around the Z-axis to create a 3D body.

Initialize the sketch design

Create a design sketch named `quarter-donut`.

```
[3]: # Initialize the donut sketch design
design = m.create_design("quarter-donut")
```

Define circle parameters

Set `path_radius`, which represents the radius of the circular path that the profile circle sweeps along, to 5 units. Set `profile_radius`, which represents the radius of the profile circle that sweeps along the path to create the donut body, to 2 units.

```
[4]: # Donut parameters
path_radius = 5
profile_radius = 2
```


Create the profile circle

Create a circle on the XZ plane centered at the coordinates (5, 0, 0) and use `profile_radius` to define the radius. This circle serves as the profile or cross-sectional shape of the donut.

```
[5]: # Create the circular profile on the XZ-plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(
    origin=Point3D([path_radius, 0, 0]),
    direction_x=UNITVECTOR3D_X,
    direction_y=UNITVECTOR3D_Z,
)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([0, 0]), profile_radius)

profile.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Perform the revolve operation

Revolve the profile circle around the Z axis to create a quarter of a donut body. Set the angle of revolution to 90 degrees in the default direction, which is counterclockwise.

```
[6]: # Revolve the profile around the Z axis and center in the absolute origin
# for an angle of 90 degrees
design.revolve_sketch(
    "quarter-donut-body",
    sketch=profile,
    axis=UNITVECTOR3D_Z,
    angle=Angle(90, unit=UNITS.degrees),
    rotation_origin=Point3D([0, 0, 0]),
)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Perform a revolve operation with a negative angle of revolution

You can use a negative angle of revolution to create a quarter of a donut in the opposite direction. The following code snippet shows how to create a quarter of a donut in the clockwise direction. The same profile circle is used, but the angle of revolution is set to -90 degrees.

```
[7]: # Initialize the donut sketch design
design = m.create_design("quarter-donut-negative")

# Revolve the profile around the Z axis and center in the absolute origin
# for an angle of -90 degrees (clockwise)
design.revolve_sketch(
    "quarter-donut-body-negative",
    sketch=profile,
```

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```

axis=UNITVECTOR3D_Z,
angle=Angle(-90, unit=UNITS.degrees),
rotation_origin=Point3D([0, 0, 0]),
)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...

```

Close the modeler

```
[8]: # Close the modeler
m.close()
```

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8.3.10 Modeling: Exporting designs

After creating a design, you typically want to bring it into a CAD tool for further development. This notebook demonstrates how to export a design to the various supported CAD formats.

Create a design

The code creates a simple design for demonstration purposes. The design consists of a set of rectangular pads with a circular hole in the center.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, UNITVECTOR3D_Y, Point2D
from ansys.geometry.core.sketch import Sketch

# Instantiate the modeler
modeler = launch_modeler()

# Create a design
design = modeler.create_design("ExportDesignExample")

# Create a sketch
sketch = Sketch()
```

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```

# Create a simple rectangle
sketch.box(Point2D([0, 0]), 10, 5)

# Extrude the sketch and displace the resulting body
# to make a plane of rectangles
for x_step in [-60, -45, -30, -15, 0, 15, 30, 45, 60]:
    for y_step in [-40, -30, -20, -10, 0, 10, 20, 30, 40]:
        # Extrude the sketch
        body = design.extrude_sketch(f"Body_X_{x_step}_Y_{y_step}", sketch, 5)

        # Displace the body in the x and y directions
        body.translate(UNITVECTOR3D_X, x_step)
        body.translate(UNITVECTOR3D_Y, y_step)

# Plot the design
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...

```

Export the design

You can export the design to various CAD formats. For the formats supported see the [DesignFileFormat](#) class, which is part of the the design module documentation.

Nonetheless, there are a set of convenience methods that you can use to export the design to the supported formats. The following code snippets demonstrate how to do it. You can decide whether to export the design to a file in a certain directory or in the current working directory.

Export to a file in the current working directory

```

[2]: # Export the design to a file in the current working directory
file_location = design.export_to_sdocx()
print(f"Design exported to {file_location}")

Design exported to C:\Users\Public\actions-runner\_work\pyansys-geometry\pyansys-
↳geometry\doc\source\examples\03_modeling\ExportDesignExample.sdocx

```

Export to a file in a certain directory

```

[3]: from pathlib import Path

# Define a downloads directory
download_dir = Path.cwd() / "downloads"

# Export the design to a file in a certain directory
file_location = design.export_to_sdocx(download_dir)
print(f"Design exported to {file_location}")

Design exported to C:\Users\Public\actions-runner\_work\pyansys-geometry\pyansys-
↳geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.sdocx

```

Export to SCDOCX format

```
[4]: # Export the design to a file in the requested directory
file_location = design.export_to_sdocx(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.sdocx
Does the file exist? True

Export to Parasolid text format

```
[5]: # Export the design to a file in the requested directory
file_location = design.export_to_parasolid_text(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.xmt_txt
Does the file exist? True

Export to Parasolid binary format

```
[6]: # Export the design to a file in the requested directory
file_location = design.export_to_parasolid_bin(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.xmt_bin
Does the file exist? True

Export to STEP format

```
[7]: # Export the design to a file in the requested directory
file_location = design.export_to_step(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.stp
Does the file exist? True

Export to IGES format

```
[8]: # Export the design to a file in the requested directory
file_location = design.export_to_iges(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.igs
Does the file exist? True

Export to FMD format

```
[9]: # Export the design to a file in the requested directory
file_location = design.export_to_fmd(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.fmd
Does the file exist? True

Export to PMDB format

```
[10]: # Export the design to a file in the requested directory
file_location = design.export_to_pmdb(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\Users\Public\actions-runner_work\pyansys-geometry\pyansys-
↳ geometry\doc\source\examples\03_modeling\downloads\ExportDesignExample.pmdb
Does the file exist? True

Close the modeler

Close the modeler after exporting the design.

```
[11]: # Close the modeler
modeler.close()
```

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8.3.11 Modeling: Visualization of the design tree on terminal

A user can visualize its model object tree easily by using the `tree_print()` method available on the Design and Component objects. This method prints the tree structure of the model in the terminal.

Perform required imports

For the following example, you need to import these modules:

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math.constants import UNITVECTOR3D_X, UNITVECTOR3D_Y
from ansys.geometry.core.math.point import Point2D, Point3D
from ansys.geometry.core.misc.units import UNITS
from ansys.geometry.core.sketch.sketch import Sketch
```

Create a design

The following code creates a simple design for demonstration purposes. The design consists of several cylinders extruded. The interesting part is visualizing the corresponding design tree.

```
[2]: # Create a modeler object
modeler = launch_modeler()

# Create your design on the server side
design = modeler.create_design("TreePrintComponent")

# Create a Sketch object and draw a circle (all client side)
sketch = Sketch()
sketch.circle(Point2D([-30, -30]), 10 * UNITS.m)
distance = 30 * UNITS.m

# The following component hierarchy is made
#
#      |----> comp_1 ---|----> nested_1_comp_1 ----> nested_1_nested_1_comp_1
#      |
#      |
#      |----> nested_2_comp_1
#      |
# DESIGN ---|----> comp_2 -----> nested_1_comp_2
#      |
#      |
#      |----> comp_3
#
#
# Now, only "comp_3", "nested_2_comp_1" and "nested_1_nested_1_comp_1"
# has a body associated.
```

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```

#

# Create the components
comp_1 = design.add_component("Component_1")
comp_2 = design.add_component("Component_2")
comp_3 = design.add_component("Component_3")
nested_1_comp_1 = comp_1.add_component("Nested_1_Component_1")
nested_1_nested_1_comp_1 = nested_1_comp_1.add_component("Nested_1_Nested_1_Component_1")
nested_2_comp_1 = comp_1.add_component("Nested_2_Component_1")
nested_1_comp_2 = comp_2.add_component("Nested_1_Component_2")

# Create the bodies
b1 = comp_3.extrude_sketch(name="comp_3_circle", sketch=sketch, distance=distance)
b2 = nested_2_comp_1.extrude_sketch(
    name="nested_2_comp_1_circle", sketch=sketch, distance=distance
)
b2.translate(UNITVECTOR3D_X, 50)
b3 = nested_1_nested_1_comp_1.extrude_sketch(
    name="nested_1_nested_1_comp_1_circle", sketch=sketch, distance=distance
)
b3.translate(UNITVECTOR3D_Y, 50)

# Create beams (in design)
circle_profile_1 = design.add_beam_circular_profile(
    "CircleProfile1", Quantity(10, UNITS.mm), Point3D([0, 0, 0]), UNITVECTOR3D_X,
    ↪UNITVECTOR3D_Y
)
beam_1 = nested_1_comp_2.create_beam(
    Point3D([9, 99, 999], UNITS.mm), Point3D([8, 88, 888], UNITS.mm), circle_profile_1
)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
↪http-equiv=&quot;Content-...

```

Visualize the design tree

Now, let's visualize the design tree using the `tree_print()` method. Let's start by printing the tree structure of the design object with no extra arguments.

```

[3]: design.tree_print()

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1

```

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(continued from previous page)

```

: |---(comp) Nested_1_Component_1
: : |---(comp) Nested_1_Nested_1_Component_1
: : |---(body) nested_1_nested_1_comp_1_circle
: |---(comp) Nested_2_Component_1
: |---(body) nested_2_comp_1_circle
|---(comp) Component_2
: |---(comp) Nested_1_Component_2
: |---(beam) 0:215
|---(comp) Component_3
|---(body) comp_3_circle

```

Controlling the depth of the tree

The `tree_print()` method accepts an optional argument `depth` to control the depth of the tree to be printed. The default value is `None`, which means the entire tree is printed.

```

[4]: design.tree_print(depth=1)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3

```

In this case, only the first level of the tree is printed - that is, the three main components.

Excluding bodies, components, or beams

By default, the `tree_print()` method prints all the bodies, components, and beams in the design tree. However, you can exclude any of these by setting the corresponding argument to `False`.

```

[5]: design.tree_print(consider_bodies=False, consider_beams=False)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
: |---(comp) Nested_1_Component_1
: : |---(comp) Nested_1_Nested_1_Component_1
: |---(comp) Nested_2_Component_1

```

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```
|---(comp) Component_2
: |---(comp) Nested_1_Component_2
|---(comp) Component_3
```

In this case, the bodies and beams are not be printed in the tree structure.

```
[6]: design.tree_print(consider_comps=False)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
```

In this case, the components are not be printed in the tree structure - leaving only the design object represented.

Sorting the tree

By default, the tree structure is sorted by the way the components, bodies, and beams were created. However, you can sort the tree structure by setting the `sort_keys` argument to `True`. In that case, the tree is sorted alphabetically.

Let's add a new component to the design and print the tree structure by default.

```
[7]: comp_4 = design.add_component("A_Component")
      design.tree_print(depth=1)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3
|---(comp) A_Component
```

Now, let's print the tree structure with the components sorted alphabetically.

```
[8]: design.tree_print(depth=1, sort_keys=True)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
```

(continues on next page)

(continued from previous page)

```

-----
(comp) TreePrintComponent
|---(comp) A_Component
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3

```

Indenting the tree

By default, the tree structure is printed with an indentation level of 4. However, you can indent the tree structure by setting the `indent` argument to the desired value.

```

[9]: design.tree_print(depth=1, indent=8)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|----- (comp) Component_1
|----- (comp) Component_2
|----- (comp) Component_3
|----- (comp) A_Component

```

In this case, the tree structure is printed with an indentation level of 8.

Printing the tree from a specific component

You can print the tree structure from a specific component by calling the `tree_print()` method on the component object.

```

[10]: nested_1_comp_1.tree_print()

>>> Tree print view of component 'Nested_1_Component_1'

Location
-----
TreePrintComponent > Component_1 > Nested_1_Component_1

Subtree
-----
(comp) Nested_1_Component_1
|---(comp) Nested_1_Nested_1_Component_1
|---(body) nested_1_nested_1_comp_1_circle

```

Closing the modeler

Finally, close the modeler.

```
[11]: modeler.close()
```

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8.3.12 Modeling: Body color assignment and usage

In PyAnsys Geometry, a *body* represents solids or surfaces organized within the *Design* assembly. As users might be already familiar with, Ansys CAD products (like SpaceClaim, Ansys Discovery and the Geometry Service), allow to assign colors to bodies. This example shows how to assign colors to a body, retrieve their value and how to use them in the client-side visualization.

Perform required imports

Perform the required imports.

```
[1]: import ansys.geometry.core as pyansys_geometry

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D, UNITVECTOR3D_X, UNITVECTOR3D_Y
from ansys.geometry.core.sketch import Sketch
```

Create a box sketch

Create a *Sketch* instance and insert a box sketch with a width and height of 10 in the default plane.

```
[2]: sketch = Sketch()
sketch.box(Point2D([0, 0]), 10, 10)

[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x223f37277d0>
```

Initiate design on server

Establish a server connection and initiate a design on the server.

```
[3]: modeler = launch_modeler()
design = modeler.create_design("ServiceColors")
```

Extrude the box sketch to create the matrix style design

Given the initial sketch, you can extrude it to create a matrix style design. In this example, you can create a 2x3 matrix of bodies. Each body is separated by 30 units in the X direction and 30 units in the Y direction. You have a total of 6 bodies.

```
[4]: translate = [[0, 30, 60], [0, 30, 60]]

for r_idx, row in enumerate(translate):
    comp = design.add_component(f"Component{r_idx}")

    for b_idx, dist in enumerate(row):
        body = comp.extrude_sketch(f"Component{r_idx}_Body{b_idx}", sketch, distance=10)
        body.translate(UNITVECTOR3D_Y, r_idx*30)
        body.translate(UNITVECTOR3D_X, dist)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv=&quot;Content-...
```

Assign colors to the bodies

Given the previous design, you can assign a color to each body. You could have done this assignment while creating the bodies, but for the sake of encapsulating the color assignment logic, it is done in its own code cell.

```
[5]: colors = ["red", "blue", "yellow"], ["orange", "green", "purple"]

for c_idx, comp in enumerate(design.components):
    for b_idx, body in enumerate(comp.bodies):
        body.color = colors[c_idx][b_idx]
        print(f"Body {body.name} has color {body.color}")
```

```
Body Component0_Body0 has color #ff0000
Body Component0_Body1 has color #0000ff
Body Component0_Body2 has color #ffff00
Body Component1_Body0 has color #ffa500
Body Component1_Body1 has color #008000
Body Component1_Body2 has color #800080
```

Plotting the design with colors

By default, the plot method does **not** use the colors assigned to the bodies. To plot the design with the assigned colors, you need to specifically request it.

Users have two options for plotting with the assigned colors:

- Pass the parameter `use_service_colors=True` to the plot method.
- Set the global parameter `USE_SERVICE_COLORS` to `True`.

It is important to note that the usage of colors when plotting might slow down the plotting process, as it requires additional information to be sent from the server to the client and processed in the client side.

If you just request the plot without setting the global parameter, the plot will be displayed without the colors, as shown below.

```
[6]: design.plot()
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

As stated previously, if you pass the parameter `use_service_colors=True` to the plot method, the plot is displayed with the assigned colors.

```
[7]: design.plot(use_service_colors=True)
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

However, if you set the global parameter to `True`, the plot is displayed with the assigned colors without the need to pass the parameter to the plot method.

```
[8]: import ansys.geometry.core as pyansys_geometry

pyansys_geometry.USE_SERVICE_COLORS = True

design.plot()

# Reverting the global parameter to its default value
pyansys_geometry.USE_SERVICE_COLORS = False

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

This last method is useful when the user wants to plot all the designs with the assigned colors without the need to pass the parameter to the plot method in every call.

Plotting specific bodies or components with colors

If the user wants to plot specific bodies with the assigned colors, the user can follow the same approach as before. The user can pass the parameter `use_service_colors=True` to the plot method or set the global parameter `USE_SERVICE_COLORS` to `True`.

In the following examples, you are shown how to do this using the `use_service_colors=True` parameter.

Let's plot the first body of the first component with the assigned colors.

```
[9]: body = design.components[0].bodies[0]

body.plot(use_service_colors=True)

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

Now, let's plot the second component with the assigned colors.

```
[10]: comp = design.components[1]

comp.plot(use_service_colors=True)

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv=&quot;Content-...
```

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8.3.13 Modeling: Surface bodies and trimmed surfaces

This example shows how to trim different surfaces, and how to use those surfaces to create surface bodies.

Create a surface

Create a sphere surface. This can be done without launching the modeler.

```
[1]: from ansys.geometry.core.shapes-surfaces import Sphere
      from ansys.geometry.core.math import Point3D

      surface = Sphere(origin=Point3D([0, 0, 0]), radius=1)
```

Now get information on how the surface is defined and parameterized.

```
[2]: surface.parameterization()
[2]: (Parameterization(form=ParamForm.PERIODIC, type=ParamType.CIRCULAR,
↳ interval=Interval(start=0, end=6.283185307179586)),
      Parameterization(form=ParamForm.CLOSED, type=ParamType.OTHER, interval=Interval(start=-
↳ 1.5707963267948966, end=1.5707963267948966)))
```

Trim the surface

For a sphere, its parametrization is (u: $[0, 2\pi]$, v: $[-\pi/2, \pi/2]$), where u corresponds to longitude and v corresponds to latitude. You can **trim** a surface by providing new parameters.

```
[3]: from ansys.geometry.core.shapes-box_uv import BoxUV
      from ansys.geometry.core.shapes-parameterization import Interval
      import math

      trimmed_surface = surface.trim(BoxUV(range_u=Interval(0, math.pi), range_v=Interval(0,
↳ math.pi/2)))
```

From a TrimmedSurface, you can always refer back to the underlying Surface if needed.

```
[4]: trimmed_surface.geometry
[4]: <ansys.geometry.core.shapes-surfaces.sphere.Sphere at 0x2533b14eb70>
```

Create a surface body

Now create a surface body by launching the modeler session and providing the trimmed surface. Then plot the body to see how you created a quarter of a sphere as a surface body.

```
[5]: from ansys.geometry.core import launch_modeler
```

```
modeler = launch_modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x2537baec830)
```

```
Ansys Geometry Modeler Client (0x2537baec7d0)
  Target:      localhost:700
  Connection: Healthy
```

```
[6]: design = modeler.create_design("SurfaceBodyExample")
body = design.create_body_from_surface("trimmed_sphere", trimmed_surface)
design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv="Content-
```

If the sphere was left untrimmed, it would create a solid body since the surface is fully closed. In this case, since the surface was open, it created a surface body.

This same process can be used with other surfaces including:

- Cone
- Cylinder
- Plane
- Torus

Each surface has its own unique parameterization, which must be understood before trying to trim it.

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[7]: modeler.close()
```

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8.3.14 Modeling: Using design parameters

You can read and update parameters that are part of the design. The simple design in this example has two associated parameters.

Perform required imports

```
[1]: from pathlib import Path
import requests

from ansys.geometry.core import launch_modeler
```

The file for this example is in the integration tests folder and can be downloaded.

Download the example file

Download the file for this example from the integration tests folder in the PyAnsys Geometry repository.

```
[2]: def download_file(url, filename):
    """Download a file from a URL and save it to a local file."""
    response = requests.get(url)
    response.raise_for_status() # Check if the request was successful
    with open(filename, 'wb') as file:
        file.write(response.content)

# URL of the file to download
url = "https://raw.githubusercontent.com/ansys/pyansys-geometry/main/tests/integration/
↳files/blockswithparameters.dsco"

# Local path to save the file to
file_path = Path.cwd() / "blockswithparameters.dsco"

# Download the file
download_file(url, file_path)
print(f"File is downloaded to {file_path}")
```

```
File is downloaded to C:\Users\Public\actions-runner\_work\pyansys-geometry\pyansys-
↳geometry\doc\source\examples\03_modeling\blockswithparameters.dsco
```

Import a design with parameters

Import the model using the `open_file()` method of the modeler.

```
[3]: # Create a modeler object
modeler = launch_modeler()
design = modeler.open_file(file_path)
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↳http-equiv="Content-...
```


Read existing parameters of the design

You can get all the parameters of the design as a list of parameters. Because this example has two parameters, you see two items in the list.

```
[4]: my_parameters = design.parameters
      print(len(my_parameters))

2
```

A parameter object has a name, value, and unit.

```
[5]: print(my_parameters[0].name)
      print(my_parameters[0].dimension_value)
      print(my_parameters[0].dimension_type)

      print(my_parameters[1].name)
      print(my_parameters[1].dimension_value)
      print(my_parameters[1].dimension_type)

p1
0.00010872999999999982
ParameterType.DIMENSIONTYPE_AREA
p2
0.0002552758322160814
ParameterType.DIMENSIONTYPE_AREA
```

Parameter values are returned in the default unit for each dimension type. Since default length unit is meter and default area unit is meter square, the value is returned in square meters.

Edit a parameter value

You can edit the parameter's name or value by simply setting these fields. Set the second parameter (p2 value to 350 mm).

```
[6]: parameter1 = my_parameters[1]
      parameter1.dimension_value = 0.000440
      response = design.set_parameter(parameter1)
      print(response)
      print(my_parameters[0].dimension_value)
      print(my_parameters[1].dimension_value)

ParameterUpdateStatus.SUCCESS
0.00010872999999999982
0.00044
```

After a successful parameter update, the design is updated. If we request the design plot again, we see the updated design.

```
[7]: design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↵http-equiv=&quot;Content-...
```

The `set_parameter()` method returns a `Success` status message if the parameter is updated or a `FAILURE` status message if the update fails. If the p2 parameter depends on the p1 parameter, updating the p1 parameter might also change the p2 parameter. In such cases, the method returns `CONSTRAINED_PARAMETERS`, which indicates other parameters were also updated.

```
[8]: parameter1 = my_parameters[0]
parameter1.dimension_value = 0.000250
response = design.set_parameter(parameter1)
print(response)
```

```
ParameterUpdateStatus.CONSTRAINED_PARAMETERS
```

To get the updated list, query the parameters once again.

```
[9]: my_parameters = design.parameters
print(my_parameters[0].dimension_value)
print(my_parameters[1].dimension_value)
```

```
0.00024999999999997274
0.0005812699999999445
```

Close the modeler

Close the modeler to free up resources and release the connection.

```
[10]: modeler.close()
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

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8.3.15 Modeling: Chamfer edges and faces

A chamfer is an angled cut on an edge. Chamfers can be created using the `Modeler.geometry_commands` module.

Create a block

Launch the modeler and create a block.

```
[1]: from ansys.geometry.core import launch_modeler, Modeler
```

```
modeler = Modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x1e96df225d0)

Ansys Geometry Modeler Client (0x1e911179940)
  Target:      127.0.0.1:700
  Connection: Healthy
```

```
[2]: from ansys.geometry.core.sketch import Sketch
      from ansys.geometry.core.math import Point2D

      design = modeler.create_design("chamfer_block")
      body = design.extrude_sketch("block", Sketch().box(Point2D([0, 0]), 1, 1), 1)

      body.plot()

      EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
      ↪http-equiv="Content-...
```

Chamfer edges

Create a uniform chamfer on all edges of the block.

```
[3]: modeler.geometry_commands.chamfer(body.edges, distance=0.1)

      body.plot()

      EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
      ↪http-equiv="Content-...
```

Chamfer faces

The chamfer of a face can also be modified. Create a chamfer on a single edge and then modify the chamfer distance value by providing the newly created face that represents the chamfer.

```
[4]: body = design.extrude_sketch("box", Sketch().box(Point2D([0,0]), 1, 1), 1)

      modeler.geometry_commands.chamfer(body.edges[0], distance=0.1)

      body.plot()

      EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
      ↪http-equiv="Content-...
```

```
[5]: modeler.geometry_commands.chamfer(body.faces[-1], distance=0.3)

      body.plot()

      EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
      ↪http-equiv="Content-...
```

Download this example

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8.4 Applied examples

These examples demonstrate the usage of PyAnsys Geometry for real-world applications.

i Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.4.1 Applied: Create a NACA 4-digit airfoil

NACA airfoils are a series of airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). They are a standardized system of airfoil shapes that are defined by a series of digits. The digits, which indicate the shape of the airfoil, are used to create the airfoil shape.

Each digit in the NACA airfoil number has a specific meaning:

- The first digit defines the maximum camber as a percentage of the chord length.
- The second digit defines the position of the maximum camber as a percentage of the chord length.
- The last two digits define the maximum thickness of the airfoil as a percentage of the chord length.

To fully understand the previous definitions, it is important to know that the chord length is the length of the airfoil from the leading edge to the trailing edge. The camber is the curvature of the airfoil, and the thickness is the distance between the upper and lower surfaces.

Symmetric airfoils have a camber of 0% and consequently, the first two digits of the NACA number are 0. For example, the NACA 0012 airfoil is a symmetric airfoil with a maximum thickness of 12%.

Define the NACA 4-digit airfoil equation

The following code uses the equation for a NACA 4-digit airfoil to create a set of points that define the airfoil shape. These points are `Point2D` objects in PyAnsys Geometry.

```
[1]: from typing import List, Union

import numpy as np

from ansys.geometry.core.math import Point2D

def naca_airfoil_4digits(number: Union[int, str], n_points: int = 200) -> list[Point2D]:
    """
    Generate a NACA 4-digits airfoil.

    Parameters
    -----
    number : int or str
        NACA 4-digit number.
    n_points : int
        Number of points to generate the airfoil. The default is ``200``.
        Number of points in the upper side of the airfoil.
        The total number of points is ``2 * n_points - 1``.

    Returns
    -----
    list[Point2D]
        List of points that define the airfoil.
    """
```

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```

# Check if the number is a string
if isinstance(number, str):
    number = int(number)

# Calculate the NACA parameters
m = number // 1000 * 0.01
p = number // 100 % 10 * 0.1
t = number % 100 * 0.01

# Generate the airfoil
points = []
for i in range(n_points):

    # Make it a exponential distribution so the points are more concentrated
    # near the leading edge
    x = (1 - np.cos(i / (n_points - 1) * np.pi)) / 2

    # Check if it is a symmetric airfoil
    if p == 0 and m == 0:
        # Camber line is zero in this case
        yc = 0
        dyc_dx = 0
    else:
        # Compute the camber line
        if x < p:
            yc = m / p**2 * (2 * p * x - x**2)
            dyc_dx = 2 * m / p**2 * (p - x)
        else:
            yc = m / (1 - p) ** 2 * ((1 - 2 * p) + 2 * p * x - x**2)
            dyc_dx = 2 * m / (1 - p) ** 2 * (p - x)

    # Compute the thickness
    yt = 5 * t * (0.2969 * x**0.5
                 - 0.1260 * x
                 - 0.3516 * x**2
                 + 0.2843 * x**3
                 - 0.1015 * x**4)

    # Compute the angle
    theta = np.arctan(dyc_dx)

    # Compute the points (upper and lower side of the airfoil)
    xu = x - yt * np.sin(theta)
    yu = yc + yt * np.cos(theta)
    xl = x + yt * np.sin(theta)
    yl = yc - yt * np.cos(theta)

    # Append the points
    points.append(Point2D([xu, yu]))
    points.insert(0, Point2D([xl, yl]))

    # Remove the first point since it is repeated

```

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```

    if i == 0:
        points.pop(0)

    return points

```

Example of a symmetric airfoil: NACA 0012

Now that the function for generating a NACA 4-digit airfoil is generated, this code creates a NACA 0012 airfoil, which is symmetric. This airfoil has a maximum thickness of 12%. The NACA number is a constant.

```
[2]: NACA_AIRFOIL = "0012"
```

Required imports

Before you start creating the airfoil points, you must import the necessary modules to create the airfoil using PyAnsys Geometry.

```
[3]: from ansys.geometry.core import launch_modeler
     from ansys.geometry.core.sketch import Sketch

```

Generate the airfoil points

Using the function defined previously, you generate the points that define the NACA 0012 airfoil. Create a Sketch object and add the points to it. Then, approximate the airfoil using straight lines between the points.

```
[4]: # Create a sketch
     sketch = Sketch()

     # Generate the points of the airfoil
     points = naca_airfoil_4digits(NACA_AIRFOIL)

     # Create the segments of the airfoil
     for i in range(len(points) - 1):
         sketch.segment(points[i], points[i + 1])

     # Close the airfoil
     sketch.segment(points[-1], points[0])

     # Plot the airfoil
     sketch.plot()

     EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
     ↪http-equiv="Content-...
```

Create the 3D airfoil

Once the Sketch object is created, you create a 3D airfoil. For this operation, you must create a modeler object, create a design object, and extrude the sketch.

```
[5]: # Launch the modeler
     modeler = launch_modeler()

```

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```

# Create the design
design = modeler.create_design(f"NACA_Airfoil_{NACA_AIRFOIL}")

# Extrude the airfoil
design.extrude_sketch("Airfoil", sketch, 1)

# Plot the design
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...

```

Save the design

Finally, save the design to a file. This file can be used in other applications or imported into a simulation software. This code saves the design as an FMD file, which can then be imported into Ansys Fluent.

```

[6]: # Save the design
file = design.export_to_fmd()
print(f"Design saved to {file}")

Design saved to C:\Users\Public\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↪doc\source\examples\04_applied\NACA_Airfoil_0012.fmd

```

Close the modeler

```

[7]: modeler.close()

```

Example of a cambered airfoil: NACA 6412

This code creates a NACA 6412 airfoil, which is cambered. This airfoil has a maximum camber of 6% and a maximum thickness of 12%. After overriding the NACA number, the code generates the airfoil points.

```

[8]: NACA_AIRFOIL = "6412"

```

Generate the airfoil points

As before, you generate the points that define the NACA 6412 airfoil. Create a `Sketch` object and add the points to it. Then, approximate the airfoil using straight lines.

```

[9]: # Create a sketch
sketch = Sketch()

# Generate the points of the airfoil
points = naca_airfoil_4digits(NACA_AIRFOIL)

# Create the segments of the airfoil
for i in range(len(points) - 1):
    sketch.segment(points[i], points[i + 1])

# Close the airfoil

```

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```

sketch.segment(points[-1], points[0])

# Plot the airfoil
sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv="Content-...
```

Create the 3D airfoil

```

[10]: # Launch the modeler
modeler = launch_modeler()

# Create the design
design = modeler.create_design(f"NACA_Airfoil_{NACA_AIRFOIL}")

# Extrude the airfoil
design.extrude_sketch("Airfoil", sketch, 1)

# Plot the design
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv="Content-...
```

Save the design

In this case, the design is saved as an SCDOCX file.

```

[11]: # Save the design
file = design.export_to_sdocx()
print(f"Design saved to {file}")

Design saved to C:\Users\Public\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↵doc\source\examples\04_applied\NACA_Airfoil_6412.sdocx
```

Close the modeler

```

[12]: modeler.close()
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.4.2 Applied: Prepare a NACA airfoil for a Fluent simulation

Once a NACA airfoil is designed, it is necessary to prepare the geometry for a CFD simulation. This notebook demonstrates how to prepare a NACA 6412 airfoil for a Fluent simulation. The starting point of this example is the previously designed NACA 6412 airfoil. The airfoil was saved in an SCDOCX file, which is now imported into the notebook. The geometry is then prepared for the simulation.

In case you want to run this notebook, make sure that you have run the previous notebook to design the NACA 6412 airfoil.

Import the NACA 6412 airfoil

The following code starts up the Geometry Service and imports the NACA 6412 airfoil. The airfoil is then displayed in the notebook.

```
[1]: import os

from ansys.geometry.core import launch_modeler

# Launch the modeler
modeler = launch_modeler()

# Import the NACA 6412 airfoil
design = modeler.open_file(os.path.join(os.getcwd(), f"NACA_Airfoil_6412.scdocx"))

# Retrieve the airfoil body
airfoil = design.bodies[0]

# Display the airfoil
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv="Content-...
```

Prepare the geometry for the simulation

The current design is only composed of the airfoil. To prepare the geometry for the simulation, you must define the domain around the airfoil. The following code creates a rectangular fluid domain around the airfoil.

The airfoil has the following dimensions:

- Chord length: 1 (X-axis)
- Thickness: Depends on NACA value (Y-axis)

Define the fluid domain as a large box with these dimensions:

- Length (X-axis) - 10 times the chord length
- Width (Z-axis) - 5 times the chord length
- Height (Y-axis) - 4 times the chord length

Place the airfoil at the center of the fluid domain.

```
[2]: from ansys.geometry.core.math import Point2D, Plane, Point3D
from ansys.geometry.core.sketch import Sketch

BOX_LENGTH = 10 # X-Axis
```

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```

BOX_WIDTH = 5 # Z-Axis
BOX_HEIGHT = 4 # Y-Axis

# Create the sketch
fluid_sketch = Sketch(
    plane=Plane(origin=Point3D([0, 0, 0.5 - (BOX_WIDTH / 2)]))
)
fluid_sketch.box(
    center=Point2D([0.5, 0]),
    height=BOX_HEIGHT,
    width=BOX_LENGTH,
)

# Extrude the fluid domain
fluid = design.extrude_sketch("Fluid", fluid_sketch, BOX_WIDTH)

```

Create named selections

Named selections are used to define boundary conditions in Fluent. The following code creates named selections for the inlet, outlet, and walls of the fluid domain. The airfoil is also assigned a named selection.

The airfoil is aligned with the X axis. The inlet is located at the left side of the airfoil, the outlet is located at the right side of the airfoil, and the walls are located at the top and bottom of the airfoil. The inlet face has therefore a negative X-axis normal vector, and the outlet face has a positive X-axis normal vector. The rest of the faces, therefore, constitute the walls.

```

[3]: # Create named selections in the fluid domain (inlet, outlet, and surrounding faces)
# Add also the airfoil as a named selection
fluid_faces = fluid.faces
surrounding_faces = []
inlet_faces = []
outlet_faces = []
for face in fluid_faces:
    if face.normal().x == 1:
        outlet_faces.append(face)
    elif face.normal().x == -1:
        inlet_faces.append(face)
    else:
        surrounding_faces.append(face)

design.create_named_selection("Outlet Fluid", faces=outlet_faces)
design.create_named_selection("Inlet Fluid", faces=inlet_faces)
design.create_named_selection("Surrounding Faces", faces=surrounding_faces)
design.create_named_selection("Airfoil Faces", faces=airfoil.faces)

```

```

[3]: <ansys.geometry.core.designer.selection.NamedSelection at 0x27f5e87bc20>

```

Display the geometry

The geometry is now ready for the simulation. The following code displays the geometry in the notebook. This example uses the `GeometryPlotter` class to display the geometry for the airfoil and fluid domain in different colors with a specified opacity level.

The airfoil is displayed in green, and the fluid domain is displayed in blue with an opacity of 0.25.

```
[4]: from ansys.geometry.core.plotting import GeometryPlotter
```

```
plotter = GeometryPlotter()
plotter.plot(airfoil, color="green")
plotter.plot(fluid, color="blue", opacity=0.15)
plotter.show()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_
↪http-equiv=&quot;Content-...
```

Export the geometry

Export the geometry into a Fluent-compatible format. The following code exports the geometry into a PMDB file, which retains the named selections.

```
[5]: # Save the design
file = design.export_to_pmdb()
print(f"Design saved to {file}.")
```

```
Design saved to C:\Users\Public\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↪doc\source\examples\04_applied\NACA_Airfoil_6412.pmdb.
```

You can import the exported PMDB file into Fluent to set up the mesh and perform the simulation. For an example of how to set up the mesh and boundary conditions in Fluent, see the [Modeling External Compressible Flow](#) example in the Fluent documentation.

The main difference between the Fluent example and this geometry is the coordinate system. The Fluent example defines the airfoil in the XY plane, while this geometry defines the airfoil in the XZ plane.

Close the modeler

```
[6]: modeler.close()
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

CONTRIBUTE

Overall guidance on contributing to a PyAnsys library appears in the [Contributing](#) topic in the *PyAnsys Developer's Guide*. Ensure that you are thoroughly familiar with this guide before attempting to contribute to PyAnsys Geometry.

The following contribution information is specific to PyAnsys Geometry.

9.1 Clone the repository

To clone and install the latest PyAnsys Geometry release in development mode, run these commands:

```
git clone https://github.com/ansys/pyansys-geometry
cd pyansys-geometry
python -m pip install --upgrade pip
pip install -e .
```

9.2 Post issues

Use the [PyAnsys Geometry Issues](#) page to submit questions, report bugs, and request new features. When possible, you should use these issue templates:

- Bug, problem, error: For filing a bug report
- Documentation error: For requesting modifications to the documentation
- Adding an example: For proposing a new example
- New feature: For requesting enhancements to the code

If your issue does not fit into one of these template categories, you can click the link for opening a blank issue.

To reach the project support team, email pyansys.core@ansys.com.

9.3 View documentation

Documentation for the latest stable release of PyAnsys Geometry is hosted at [PyAnsys Geometry Documentation](#).

In the upper right corner of the documentation's title bar, there is an option for switching from viewing the documentation for the latest stable release to viewing the documentation for the development version or previously released versions.

9.4 Adhere to code style

PyAnsys Geometry follows the PEP8 standard as outlined in [PEP 8](#) in the *PyAnsys Developer's Guide* and implements style checking using `pre-commit`.

To ensure your code meets minimum code styling standards, run these commands:

```
pip install pre-commit
pre-commit run --all-files
```

You can also install this as a pre-commit hook by running this command:

```
pre-commit install
```

This way, it's not possible for you to push code that fails the style checks:

```
$ pre-commit install
$ git commit -am "added my cool feature"
ruff.....Passed
codespell.....Passed
check for merge conflicts.....Passed
debug statements (python).....Passed
check yaml.....Passed
trim trailing whitespace.....Passed
Validate GitHub Workflows.....Passed
check pre-commit.ci config.....Passed
```

9.5 Build the documentation

Note

To build the documentation, you must have the Geometry Service installed and running on your machine because it is used to generate the examples in the documentation. It is also recommended that the service is running as a Docker container.

If you do not have the Geometry Service installed, you can still build the documentation, but the examples are not generated. To build the documentation without the examples, define the following environment variable:

```
# On Linux or macOS
export BUILD_EXAMPLES=false

# On Windows CMD
set BUILD_EXAMPLES=false

# On Windows PowerShell
$env:BUILD_EXAMPLES="false"
```

To build the documentation locally, you must run this command to install the documentation dependencies:

```
pip install -e .[doc]
```

Then, navigate to the docs directory and run this command:

```
# On Linux or macOS
make html

# On Windows
./make.bat html
```

The documentation is built in the docs/_build/html directory.

You can clean the documentation build by running this command:

```
# On Linux or macOS
make clean

# On Windows
./make.bat clean
```

9.6 Adding examples

Users can collaborate with examples to this documentation by adding new examples. A reference commit of the changes that adding an example requires is shown here:

<https://github.com/ansys/pyansys-geometry/pull/1454/commits/7fcf02f86f05e0e5ce1c1071c3c5fcd274ec481c>

To add a new example, follow these steps:

1. Create a new notebook in the doc/source/examples directory, under the appropriate folder for your example.
2. Take as reference an existing example to see how the notebook should be structured.
3. Add the new notebook to the doc/source/examples.rst file.
4. Store a thumbnail image of the example in the doc/source/_static/thumbnails directory.
5. Link the thumbnail image to your example file in doc/source/conf.py as shown in the reference commit.

You can also test the correct build process of a new example by performing the following steps:

1. Run the following command to install the documentation dependencies:

```
pip install -e .[doc]
```

2. Install myst-nb by running this command:

```
pip install myst-nb
```

3. Run the following command to build a single example (that is, substitute <PATH_TO_MY_EXAMPLE_FILE> with the path to your example file):

```
mystnb-docutils-html --nb-read-as-md=1 <PATH_TO_MY_EXAMPLE_FILE> output.html
```

4. Check the output file output.html to ensure that the example is correctly built. Rendered output does not have documentation styling but users should have the ability to see its proper execution.

Note

Plots are not be rendered in the output file, but the code and markdown cells should be correctly rendered. In case of failure during execution users can also see the error message in the output file.

9.7 Run tests

PyAnsys Geometry uses `pytest` for testing.

9.7.1 Prerequisites

Prior to running the tests, you must run this command to install the test dependencies:

```
pip install -e .[tests]
```

Make sure to define the port and host of the service using the following environment variables:

```
# On Linux or macOS
export ANSRV_GEO_PORT=5000
export ANSRV_GEO_HOST=localhost

# On Windows CMD
set ANSRV_GEO_PORT=5000
set ANSRV_GEO_HOST=localhost

# On Windows PowerShell
$env:ANSRV_GEO_PORT=5000
$env:ANSRV_GEO_HOST="localhost"
```

9.7.2 Running the tests

To run the tests, navigate to the root directory of the repository and run this command:

```
pytest
```

Note

The tests require the Geometry Service to be installed and running on your machine. The tests fail if the service is not running. It is expected for the Geometry Service to be running as a Docker container.

If you do not have the Geometry Service running as a Docker container, but you have it running on your machine, you can still run the tests with the following argument:

```
pytest --use-existing-service=yes
```

In this section, users are able to download a set of assets related to PyAnsys Geometry.

10.1 Documentation

The following links provide users with downloadable documentation in various formats

- [Documentation in HTML format](#)
- [Documentation in PDF format](#)

10.2 Wheelhouse

If you lack an internet connection on your installation machine, you should install PyAnsys Geometry by downloading the wheelhouse archive.

Each wheelhouse archive contains all the Python wheels necessary to install PyAnsys Geometry from scratch on Windows, Linux, and MacOS from Python 3.10 to 3.13. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

For example, on Linux with Python 3.10, unzip the wheelhouse archive and install it with:

```
unzip ansys-geometry-core-v0.9.0-wheelhouse-ubuntu-latest-3.10.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

If you are on Windows with Python 3.10, unzip to a wheelhouse directory by running `-d wheelhouse` (this is required for unzipping to a directory on Windows) and install using the preceding command.

Consider installing using a [virtual environment](#).

The following wheelhouse files are available for download:

10.2.1 Linux

- [Linux wheelhouse for Python 3.10](#)
- [Linux wheelhouse for Python 3.11](#)
- [Linux wheelhouse for Python 3.12](#)
- [Linux wheelhouse for Python 3.13](#)

10.2.2 Windows

- Windows wheelhouse for Python 3.10
- Windows wheelhouse for Python 3.11
- Windows wheelhouse for Python 3.12
- Windows wheelhouse for Python 3.13

10.2.3 MacOS

- MacOS wheelhouse for Python 3.10
- MacOS wheelhouse for Python 3.11
- MacOS wheelhouse for Python 3.12
- MacOS wheelhouse for Python 3.13

10.3 Geometry service Docker container assets

Build the latest Geometry service Docker container using the following assets. For information on how to build the container, see *Docker containers*.

Currently, the Geometry service backend is mainly delivered as a **Windows** Docker container. However, these containers require a Windows machine to run them.

10.3.1 Windows container

Note

Only users with access to <https://github.com/ansys/pyansys-geometry-binaries> can download these binaries.

- Latest Geometry service binaries for Windows containers
- Latest Geometry service Dockerfile for Windows containers

RELEASE NOTES

This document contains the release notes for the PyAnsys Geometry project.

11.1 0.9.0 - 2025-02-18

11.1.1 Added

- design activation changes #1707
- add contributors #1708
- Implementation of inspect & repair geometry #1712
- launch core service from envar #1716
- workflow enhancements for better tool results #1723
- add face color, round info, bring measure tools to linux #1732
- conservative approach to single design per modeler #1740
- export glb #1741
- allow plotting of individual faces #1757

11.1.2 Dependencies

- bump ansys-api-geometry from 0.4.33 to 0.4.34 #1709
- bump ansys-sphinx-theme[autoapi] from 1.2.6 to 1.2.7 in the docs-deps group #1719
- bump ansys-api-geometry from 0.4.34 to 0.4.35 #1720
- bump ansys-sphinx-theme[autoapi] from 1.2.7 to 1.3.0 in the docs-deps group #1726
- bump ansys-sphinx-theme[autoapi] from 1.3.0 to 1.3.1 in the docs-deps group #1728
- bump ansys-api-geometry from 0.4.35 to 0.4.36 #1729
- bump trame-vtk from 2.8.14 to 2.8.15 #1736
- bump jupyter from 1.16.6 to 1.16.7 in the docs-deps group #1742
- bump ansys-api-geometry from 0.4.36 to 0.4.37 #1743
- bump myst-parser from 4.0.0 to 4.0.1 in the docs-deps group #1744
- bump ansys-api-geometry from 0.4.37 to 0.4.38 #1746
- bump numpy from 2.2.2 to 2.2.3 #1747
- bump panel from 1.6.0 to 1.6.1 #1749

- bump scipy from 1.15.1 to 1.15.2 #1756

11.1.3 Documentation

- update CONTRIBUTING.md #1730

11.1.4 Fixed

- re enable fmd tests #1711
- support body mirror on linux #1714
- use sketch plane for imprint/project curves #1715
- revert boolean ops logic and hold-off on commands-based implementation (temporarily) #1725
- Linux Core Service docker file was missing a dependency #1758

11.1.5 Maintenance

- update CHANGELOG for v0.8.2 #1706
- pre-commit automatic update #1717, #1737
- update SECURITY.md versions supported #1722
- keep simba-plugin-geometry tag #1739
- enhancements to GLB export and object plot() methods #1750
- clean up deprecation warning for trapezoid class and add more info on deprecation #1754

11.1.6 Test

- verifying issue with empty intersect and temporal body creation #1258
- Expand pattern tests #1713
- set body name #1727
- activate 8 linux tests #1745

11.2 0.8.2 - 2025-01-29

11.2.1 Added

- create a fillet on an edge/face #1621
- create a full fillet between multiple faces #1623
- extrude existing faces, setup face offset relationships #1628
- interference repair tool #1633
- extrude existing edges to create surface bodies #1638
- create and modify linear patterns #1641
- body suppression state #1643
- parameters refurbished #1647
- rename object #1648
- surface body from trimmed curves #1650

- create circular and fill patterns #1653
- find fix simplify #1661
- replace face #1664
- commands for merge and intersect #1665
- revolve faces a set distance, up to another object, or by a helix #1666
- add split body and tests #1669
- enable get/set persistent ids for stride import/export #1671
- find and fix edge methods #1672
- shell methods #1673
- implementation of NURBS curves #1675
- get assigned material #1684
- matrix rotation and translation #1689
- is_core_service BackendType static method #1692
- export and download stride format #1698
- blitz development #1701

11.2.2 Dependencies

- bump ansys-tools-visualization-interface from 0.7.0 to 0.8.1 #1640
- bump ansys-api-geometry from 0.4.27 to 0.4.28 #1644
- bump sphinx-autodoc-typehints from 3.0.0 to 3.0.1 in the docs-deps group #1651
- bump ansys-api-geometry from 0.4.28 to 0.4.30 #1652
- bump protobuf from 5.28.3 to 5.29.3 in the grpc-deps group across 1 directory #1656
- bump numpy from 2.2.1 to 2.2.2 #1662
- bump ansys-api-geometry from 0.4.30 to 0.4.31 #1663
- bump ansys api geometry from 0.4.30 to 0.4.32 #1677
- bump ansys-api-geometry from 0.4.31 to 0.4.32 #1681
- bump panel from 1.5.5 to 1.6.0 #1682
- bump semver from 3.0.2 to 3.0.4 #1687
- bump ansys-api-geometry from 0.4.32 to 0.4.33 #1695
- bump nbconvert from 7.16.5 to 7.16.6 in the docs-deps group #1700

11.2.3 Fixed

- reactivate test on failing extra edges test #1396
- filter set export id to only CoreService based backends #1685
- cleanup unsupported module #1690
- disable unimplemented tests #1691
- tech review fixes for blitz branch #1703

11.2.4 Maintenance

- update CHANGELOG for v0.8.1 #1639
- whitelist semver package temporarily #1657
- reverting semver package whitelist since problematic version is yanked #1659
- pre-commit automatic update #1667, #1696
- ensure design is closed on test exit #1680
- use dedicate pygeometry-ci-2 runner #1693
- remove towncrier info duplicates #1702

11.2.5 Test

- add more find and fix tests for repair tools #1645
- Add some new tests #1670
- add unit tests for 3 repair tools #1683

11.3 0.8.1 - 2025-01-15

11.3.1 Dependencies

- bump ansys-api-geometry from 0.4.26 to 0.4.27 #1634

11.3.2 Fixed

- release issues encountered #1637

11.3.3 Maintenance

- update CHANGELOG for v0.8.0 #1636

11.4 0.8.0 - 2025-01-15

11.4.1 Added

- active support for Python 3.13 #1481
- add chamfer tool #1495
- allow version input to automatically consider the nuances for the Ansys Student version #1548
- adapt health check timeout algorithm #1559
- add core service support #1571
- enable (partially) prepare and repair tools in Core Service #1580
- create launcher for core services #1587

11.4.2 Dependencies

- bump ansys-api-geometry from 0.4.16 to 0.4.17 #1547
- bump ansys-sphinx-theme[autoapi] from 1.2.1 to 1.2.2 in the docs-deps group #1549
- bump ansys-api-geometry from 0.4.17 to 0.4.18 #1550
- bump ansys-tools-visualization-interface from 0.5.0 to 0.6.0 #1554
- bump pytest from 8.3.3 to 8.3.4 #1562
- bump six from 1.16.0 to 1.17.0 #1568
- bump the docs-deps group across 1 directory with 2 updates #1570
- bump ansys-api-geometry from 0.4.18 to 0.4.20 #1574
- bump numpy from 2.1.3 to 2.2.0 #1575
- bump ansys-api-geometry from 0.4.20 to 0.4.23 #1581
- bump ansys-api-geometry from 0.4.23 to 0.4.24 #1582
- bump ansys-tools-visualization-interface from 0.6.0 to 0.6.1 #1583
- bump ansys-tools-visualization-interface from 0.6.1 to 0.6.2 #1586
- avoid the usage of attrs 24.3.0 (temporary) #1589
- bump jupyter from 1.16.4 to 1.16.5 in the docs-deps group #1590
- bump jupyter from 1.16.5 to 1.16.6 in the docs-deps group #1593
- bump panel from 1.5.4 to 1.5.5 #1595
- bump ansys-sphinx-theme[autoapi] from 1.2.3 to 1.2.4 in the docs-deps group #1597
- bump notebook from 7.3.1 to 7.3.2 in the docs-deps group #1598
- bump numpy from 2.2.0 to 2.2.1 #1599
- bump ansys-tools-path from 0.7.0 to 0.7.1 #1600
- bump nbsphinx from 0.9.5 to 0.9.6 in the docs-deps group #1602
- bump nbconvert from 7.16.4 to 7.16.5 in the docs-deps group #1609
- bump ansys-api-geometry from 0.4.24 to 0.4.25 #1610
- bump sphinx-autodoc-typehints from 2.5.0 to 3.0.0 in the docs-deps group #1611
- bump scipy from 1.14.1 to 1.15.0 #1612
- bump trame-vtk from 2.8.12 to 2.8.13 #1616
- bump trame-vtk from 2.8.13 to 2.8.14 #1617
- bump ansys-tools-visualization-interface from 0.6.2 to 0.7.0 #1619
- bump ansys-sphinx-theme[autoapi] from 1.2.4 to 1.2.6 in the docs-deps group #1624
- bump scipy from 1.15.0 to 1.15.1 #1625
- bump ansys-api-geometry from 0.4.25 to 0.4.26 #1626

11.4.3 Documentation

- Explain how to report a security issue. #1605

11.4.4 Fixed

- numpydoc warnings #1556
- vtk/pyvista issues #1584
- make_child_logger only takes 2 args. #1603
- FAQ on install #1631

11.4.5 Maintenance

- pre-commit automatic update #1366, #1552, #1561, #1588, #1601, #1615, #1630
- update CHANGELOG for v0.7.6 #1545
- change release artifacts self-hosted runner #1546
- automerge pre-commit.ci PRs #1553
- bump pyvista/setup-headless-display-action to v3 #1555
- decouple unstable image promotion #1591
- skip unnecessary stages when containers are the same #1592
- Numpy is already imported at the top of the module. #1604
- update license year using pre-commit hook #1608

11.5 0.7.6 - 2024-11-19

11.5.1 Added

- allow for some additional extrusion direction names #1534

11.5.2 Dependencies

- bump ansys-sphinx-theme[autoapi] from 1.1.7 to 1.2.0 in the docs-deps group #1520
- bump ansys-tools-visualization-interface from 0.4.7 to 0.5.0 #1521
- bump numpy from 2.1.2 to 2.1.3 #1522
- bump ansys-api-geometry from 0.4.13 to 0.4.14 #1525
- bump ansys-api-geometry from 0.4.14 to 0.4.15 #1529
- bump pint from 0.24.3 to 0.24.4 #1530
- bump trame-vtk from 2.8.11 to 2.8.12 #1531
- bump ansys-sphinx-theme[autoapi] from 1.2.0 to 1.2.1 in the docs-deps group #1535
- bump panel from 1.5.3 to 1.5.4 #1536
- bump ansys-tools-path from 0.6.0 to 0.7.0 #1537
- bump ansys-api-geometry from 0.4.15 to 0.4.16 #1538
- limit upper version on grpcio & grpcio-health-checking to 1.68 #1544

11.5.3 Documentation

- typo with the docstrings #1524
- change max header links before more dropdown #1527

11.5.4 Maintenance

- update CHANGELOG for v0.7.5 #1519
- pre-commit automatic update #1523, #1532, #1543
- bump codecov/codecov-action from 4 to 5 in the actions group #1541

11.6 0.7.5 - 2024-10-31

11.6.1 Added

- create body from surface #1454
- performance enhancements to plotter #1496
- allow picking from easy access methods #1499
- implement cut operation in extrude sketch #1510
- caching bodies to avoid unnecessary object creation #1513
- enable retrieval of service logs (via API) #1515

11.6.2 Dependencies

- bump sphinx from 8.1.0 to 8.1.3 in the docs-deps group #1479
- bump ansys-sphinx-theme[autoapi] from 1.1.4 to 1.1.5 in the docs-deps group #1482
- bump the grpc-deps group across 1 directory with 3 updates #1487
- bump ansys-sphinx-theme[autoapi] from 1.1.5 to 1.1.6 in the docs-deps group #1493
- bump trame-vtk from 2.8.10 to 2.8.11 #1494
- bump ansys-api-geometry from 0.4.11 to 0.4.12 #1502
- bump protobuf from 5.28.2 to 5.28.3 in the grpc-deps group #1505
- bump ansys-sphinx-theme[autoapi] from 1.1.6 to 1.1.7 in the docs-deps group #1506
- bump ansys-tools-visualization-interface from 0.4.6 to 0.4.7 #1507
- bump panel from 1.5.2 to 1.5.3 #1508
- bump ansys-api-geometry from 0.4.12 to 0.4.13 #1512
- bump the grpc-deps group with 2 updates #1517
- bump pytest-cov from 5.0.0 to 6.0.0 #1518

11.6.3 Documentation

- avoid having a drop down in the top navigation bar #1485
- provide information on how to build a single example #1490
- add example file to download in the test #1501

- revisit examples to make sure they are properly styled #1509
- align landing page layout with UI/UX requirements #1511

11.6.4 Fixed

- static search options #1478
- respect product_version when launching geometry service #1486

11.6.5 Maintenance

- update CHANGELOG for v0.7.4 #1476
- pre-commit automatic update #1480, #1516
- avoid linkcheck on changelog (unnecessary) #1489
- update CONTRIBUTORS #1492
- allowing new tags for Windows Core Service #1497
- simplify vulnerabilities check #1504

11.7 0.7.4 - 2024-10-11

11.7.1 Dependencies

- bump sphinx from 8.0.2 to 8.1.0 in the docs-deps group #1470
- bump ansys-api-geometry from 0.4.10 to 0.4.11 #1473
- bump ansys-sphinx-theme to v1.1.3 #1475

11.7.2 Fixed

- solving intersphinx warnings on paths #1469
- check_input_types not working with forward refs #1471
- share_topology is available on 24R2 #1472

11.7.3 Maintenance

- update CHANGELOG for v0.7.3 #1466

11.8 0.7.3 - 2024-10-09

11.8.1 Added

- use service colors in plotter (upon request) #1376
- capability to close designs (also on `modeler.exit()`) #1409
- prioritize user-defined SPACECLAIM_MODE env var #1440
- verifying Linux service also accepts colors #1451

11.8.2 Dependencies

- bump protobuf from 5.28.0 to 5.28.1 in the grpc-deps group #1424
- bump the docs-deps group with 2 updates #1425, #1436
- bump ansys-tools-visualization-interface from 0.4.3 to 0.4.4 #1426
- bump pytest from 8.3.2 to 8.3.3 #1427
- bump panel from 1.4.5 to 1.5.0 #1428
- bump protobuf from 5.28.1 to 5.28.2 in the grpc-deps group #1435
- bump the grpc-deps group with 3 updates #1442
- bump beartype from 0.18.5 to 0.19.0 #1443
- bump panel from 1.5.0 to 1.5.1 #1444
- bump ansys-sphinx-theme[autoapi] from 1.1.1 to 1.1.2 in the docs-deps group #1456
- bump ansys-api-geometry from 0.4.8 to 0.4.9 #1457
- bump numpy from 2.1.1 to 2.1.2 #1458
- bump panel from 1.5.1 to 1.5.2 #1459
- bump ansys-api-geometry from 0.4.9 to 0.4.10 #1461
- bump ansys-tools-visualization-interface from 0.4.4 to 0.4.5 #1462
- update protobuf from 5.27.2 to 5.27.5 #1464
- bump sphinx-autodoc-typehints from 2.4.4 to 2.5.0 in the docs-deps group #1465

11.8.3 Documentation

- adding cheat sheet on documentation #1433
- add captions in examples toctrees #1434

11.8.4 Fixed

- ci/cd issues on documentation build #1441
- adapt tessellate tests to new core service #1449
- rename folders on Linux docker image according to new version #1450

11.8.5 Maintenance

- update CHANGELOG for v0.7.2 #1422
- checkout LFS files from previous version to ensure upload #1423
- pre-commit automatic update #1431, #1437, #1445, #1460
- update to ansys actions v8 and docs theme (static search) #1446
- pyvista/setup-headless-display started failing #1447
- check method implemented in Ansys actions #1448
- unstable image promotion and dependabot daily updates #1463

11.9 0.7.2 - 2024-09-11

11.9.1 Added

- allow for platform input when using Ansys Lab #1416
- ensure GrpcClient class closure upon deletion #1417

11.9.2 Dependencies

- bump sphinx-autodoc-typehints from 2.3.0 to 2.4.0 in the docs-deps group #1411
- bump numpy from 2.1.0 to 2.1.1 #1412
- bump ansys-tools-visualization-interface from 0.4.1 to 0.4.3 #1413

11.9.3 Documentation

- remove title from landing page #1408
- adapt examples to use launch_modeler instead of Modeler obj connection #1410

11.9.4 Fixed

- handle properly `np.cross()` - 2d ops deprecated in Numpy 2.X #1419
- change logo link so that it renders properly on PyPI #1420
- wrong path on logo image #1421

11.9.5 Maintenance

- update CHANGELOG for v0.7.1 #1407
- pre-commit automatic update #1418

11.10 0.7.1 - 2024-09-06

11.10.1 Added

- get and set body color #1357
- add `modeler.exit()` method #1375
- setting instance name during component creation #1382
- accept `pathlib.Path` as input in missing methods #1385
- default logs folder on Geometry Service started by Python at PUBLIC (Windows) #1386
- allowing users to specify API version when running script against SpaceClaim or Discovery #1395
- expose `modeler.designs` attribute #1401
- pretty print components #1403

11.10.2 Dependencies

- bump the grpc-deps group with 2 updates #1363, #1369
- bump the docs-deps group with 2 updates #1364, #1392
- bump numpy from 2.0.1 to 2.1.0 #1365
- bump ansys-sphinx-theme[autoapi] from 1.0.5 to 1.0.7 in the docs-deps group #1370
- bump ansys-api-geometry from 0.4.7 to 0.4.8 #1371
- bump scipy from 1.14.0 to 1.14.1 #1372
- bump the grpc-deps group with 3 updates #1391
- bump ansys-tools-visualization-interface from 0.4.0 to 0.4.1 #1393
- bump ansys-sphinx-theme[autoapi] from 1.0.7 to 1.0.8 in the docs-deps group #1397

11.10.3 Documentation

- add project logo #1405

11.10.4 Fixed

- remove server_logs_folder argument for Discovery and SpaceClaim #1387

11.10.5 Maintenance

- update CHANGELOG for v0.7.0 #1360
- bump dev branch to v0.8.dev0 #1361
- solving various warnings #1368
- pre-commit automatic update #1373, #1394
- upload coverage artifacts properly with upload-artifact@v4.4.0 #1406

11.11 0.7.0 - 2024-08-13

11.11.1 Added

- build: drop support for Python 3.9 #1341
- feat: adapting beartype typehints to +Python 3.10 standard #1347

11.11.2 Dependencies

- build: bump the grpc-deps group with 3 updates #1342
- build: bump panel from 1.4.4 to 1.4.5 #1344
- bump the docs-deps group across 1 directory with 4 updates #1353
- bump trame-vtk from 2.8.9 to 2.8.10 #1355
- bump ansys-api-geometry from 0.4.6 to 0.4.7 #1356

11.11.3 Documentation

- feat: update conf for version 1.x of ansys-sphinx-theme #1351

11.11.4 Fixed

- trapezoid signature change and internal checks #1354

11.11.5 Maintenance

- updating Ansys actions to v7 - changelog related #1348
- ci: bump ansys/actions from 6 to 7 in the actions group #1352
- pre-commit automatic update #1358

11.11.6 Miscellaneous

- chore: pre-commit automatic update #1345

11.12 0.6.6 - 2024-08-01

11.12.1 Added

- feat: Add misc. repair and prepare tool methods #1293
- feat: name setter and fill style getter setters #1299
- feat: extract fluid volume from solid #1306
- feat: keep “other” bodies when performing bool operations #1311
- feat: revolve_sketch rotation definition enhancement #1336

11.12.2 Changed

- chore: update CHANGELOG for v0.6.5 #1290
- chore: enable ruff formatter on pre-commit #1312
- chore: updating dependabot groups #1313
- chore: adding issue links to TODOs #1320
- feat: adapt to new ansys-tools-visualization-interface v0.4.0 #1338

11.12.3 Fixed

- test: create sphere bug raised after box creation #1291
- ci: docker cleanup #1294
- fix: default length units not being used properly on arc creation #1310

11.12.4 Dependencies

- build: bump ansys-api-geometry from 0.4.4 to 0.4.5 #1292
- build: bump pyvista[jupyter] from 0.43.10 to 0.44.0 in the docs-deps group #1296
- build: bump jupyter from 1.16.2 to 1.16.3 in the docs-deps group #1300

- build: bump ansys-api-geometry from 0.4.5 to 0.4.6 #1301
- build: bump pint from 0.24.1 to 0.24.3 #1307
- build: bump grpcio-health-checking from 1.60.0 to 1.64.1 in the grpc-deps group #1315
- build: bump the docs-deps group across 1 directory with 2 updates #1316
- build: bump the grpc-deps group with 2 updates #1322
- build: bump the docs-deps group with 2 updates #1323
- build: bump pyvista[jupyter] from 0.44.0 to 0.44.1 #1324
- build: bump ansys-tools-visualization-interface from 0.2.6 to 0.3.0 #1325
- build: bump pytest from 8.2.2 to 8.3.1 #1326
- build: bump pytest from 8.3.1 to 8.3.2 #1331
- build: bump numpy from 2.0.0 to 2.0.1 #1332

11.12.5 Miscellaneous

- chore: pre-commit automatic update #1327, #1333

11.13 0.6.5 - 2024-07-02

11.13.1 Changed

- chore: update CHANGELOG for v0.6.4 #1278
- build: update sphinx-autodoc-typehints version #1280
- chore: update SECURITY.md #1286

11.13.2 Fixed

- fix: manifest path should render as posix rather than uri #1289

11.13.3 Dependencies

- build: bump protobuf from 5.27.1 to 5.27.2 in the grpc-deps group #1283
- build: bump scipy from 1.13.1 to 1.14.0 #1284
- build: bump vtk from 9.3.0 to 9.3.1 #1287

11.13.4 Miscellaneous

- chore: pre-commit automatic update #1281, #1288

11.14 0.6.4 - 2024-06-24

11.14.1 Added

- feat: using ruff as the main linter/formatter #1274

11.14.2 Changed

- chore: update CHANGELOG for v0.6.3 #1273
- chore: bump pre-commit-hook version #1276

11.14.3 Fixed

- fix: backticks breaking doc build after ruff linter #1275

11.14.4 Dependencies

- build: bump pint from 0.24 to 0.24.1 #1277

11.15 0.6.3 - 2024-06-18

11.15.1 Changed

- chore: update CHANGELOG for v0.6.2 #1263
- build: adapting to numpy 2.x #1265
- docs: using ansys actions (again) to build docs #1270

11.15.2 Fixed

- fix: unnecessary Point3D comparison #1264
- docs: examples are not being uploaded as assets (.py/.ipynb) #1268
- fix: change action order #1269

11.15.3 Dependencies

- build: bump numpy from 1.26.4 to 2.0.0 #1266
- build: bump the docs-deps group with 2 updates #1271

11.15.4 Miscellaneous

- chore: pre-commit automatic update #1267

11.16 0.6.2 - 2024-06-17

11.16.1 Added

- feat: deprecating log_level and logs_folder + adding client log control #1260
- feat: adding deprecation support for args and methods #1261

11.16.2 Changed

- chore: update CHANGELOG for v0.6.1 #1256
- ci: simplify doc build using ansys/actions #1262

11.16.3 Fixed

- fix: Rename built in shadowing variables #1257

11.17 0.6.1 - 2024-06-12

11.17.1 Added

- feat: revolve a sketch given an axis and an origin #1248

11.17.2 Changed

- chore: update CHANGELOG for v0.6.0 #1245
- chore: update dev version to 0.8.dev0 #1246

11.17.3 Fixed

- fix: Bug in *show* function #1255

11.17.4 Dependencies

- build: bump protobuf from 5.27.0 to 5.27.1 in the grpc-deps group #1250
- build: bump the docs-deps group with 2 updates #1251
- build: bump trame-vtk from 2.8.8 to 2.8.9 #1252
- build: bump pint from 0.23 to 0.24 #1253
- build: bump ansys-tools-visualization-interface from 0.2.2 to 0.2.3 #1254

11.17.5 Miscellaneous

- docs: add conda information for package #1247

11.18 0.6.0 - 2024-06-07

11.18.1 Added

- feat: Adapt to ansys-visualizer #959
- fix: rename `GeomPlotter` to `GeometryPlotter` #1227
- refactor: use ansys-tools-visualization-interface global vars rather than env vars #1230
- feat: bump to use v251 as default #1242

11.18.2 Changed

- chore: update CHANGELOG for v0.5.6 #1213
- chore: update SECURITY.md #1214
- ci: use Trusted Publisher for releasing package #1216
- ci: remove pygeometry-ci-1 specific logic #1221
- ci: only run doc build on runners outside the ansys network #1223
- chore: pre-commit automatic update #1224

- ci: announce nightly workflows failing #1237
- ci: failing notifications improvement #1243
- fix: broken interactive docs and improved tests paths #1244

11.18.3 Fixed

- fix: Interactive documentation #1226
- fix: only notify on failure and fill with data #1238

11.18.4 Dependencies

- build: bump protobuf from 5.26.1 to 5.27.0 in the grpc-deps group #1217
- build: bump panel from 1.4.2 to 1.4.3 in the docs-deps group #1218
- build: bump ansys-api-geometry from 0.4.1 to 0.4.2 #1219
- build: bump ansys-sphinx-theme[autoapi] from 0.16.2 to 0.16.5 in the docs-deps group #1231
- build: bump requests from 2.32.2 to 2.32.3 #1232
- build: bump ansys-api-geometry from 0.4.2 to 0.4.3 #1233
- build: bump ansys-tools-visualization-interface from 0.2.1 to 0.2.2 #1234
- build: bump panel from 1.4.3 to 1.4.4 in the docs-deps group #1235
- build: bump ansys-tools-path from 0.5.2 to 0.6.0 #1236
- build: bump grpcio from 1.64.0 to 1.64.1 in the grpc-deps group #1239
- build: bump ansys-api-geometry from 0.4.3 to 0.4.4 #1240
- build: bump pytest from 8.2.1 to 8.2.2 #1241

11.18.5 Miscellaneous

- docs: update AUTHORS #1222

11.19 0.5.6 - 2024-05-23

11.19.1 Added

- feat: add new arc constructors #1208

11.19.2 Changed

- chore: update CHANGELOG for v0.5.5 #1205

11.19.3 Dependencies

- build: bump requests from 2.31.0 to 2.32.2 #1204
- build: bump ansys-sphinx-theme[autoapi] from 0.16.0 to 0.16.2 in the docs-deps group #1210
- build: bump docker from 7.0.0 to 7.1.0 #1211
- build: bump scipy from 1.13.0 to 1.13.1 #1212

11.20 0.5.5 - 2024-05-21

11.20.1 Changed

- docs: adapt ansys_sphinx_theme_autoapi extension for autoapi #1135
- chore: update CHANGELOG for v0.5.4 #1194

11.20.2 Fixed

- fix: adapting Arc class constructor order to (start, end, center) #1196
- chore: limit requests library version under 2.32 #1203

11.20.3 Dependencies

- build: bump grpcio from 1.63.0 to 1.64.0 in the grpc-deps group #1198
- build: bump the docs-deps group with 2 updates #1199
- build: bump pytest from 8.2.0 to 8.2.1 #1200

11.20.4 Miscellaneous

- chore: pre-commit automatic update #1202

11.21 0.5.4 - 2024-05-15

11.21.1 Added

- feat: allow for product_version on geometry service launcher function #1182

11.21.2 Changed

- chore: update CHANGELOG for v0.5.3 #1177

11.21.3 Dependencies

- build: bump the docs-deps group with 4 updates #1178
- build: bump pytest from 8.1.1 to 8.2.0 #1179
- build: bump grpcio from 1.62.2 to 1.63.0 in the grpc-deps group #1186
- build: bump the docs-deps group with 2 updates #1187
- build: bump trame-vtk from 2.8.6 to 2.8.7 #1188
- build: bump nbsphinx from 0.9.3 to 0.9.4 in the docs-deps group #1189
- build: bump trame-vtk from 2.8.7 to 2.8.8 #1190

11.21.4 Miscellaneous

- chore: pre-commit automatic update #1180, #1193
- docs: add geometry preparation for Fluent simulation #1183

11.22 0.5.3 - 2024-04-29

11.22.1 Fixed

- fix: semver intersphinx mapping not resolved properly #1175
- fix: start and end points for edge #1176

11.23 0.5.2 - 2024-04-29

11.23.1 Added

- feat: add semver to intersphinx #1173

11.23.2 Changed

- chore: update CHANGELOG for v0.5.1 #1165
- chore: bump version to v0.6.dev0 #1166
- chore: update CHANGELOG for v0.5.2 #1172
- fix: allow to reuse last release binaries (if requested) #1174

11.23.3 Fixed

- fix: GetSurface and GetCurve not available prior to 24R2 #1171

11.23.4 Miscellaneous

- docs: creating a NACA airfoil example #1167
- docs: simplify README example #1169

11.24 0.5.1 - 2024-04-24

11.24.1 Added

- feat: security updates dropped for v0.3 or earlier #1126
- feat: add `export_to` functions #1147

11.24.2 Changed

- ci: adapt to vale v3 #1129
- ci: bump ansys/actions from 5 to 6 in the actions group #1133
- docs: add release notes in our documentation #1138
- chore: bump ansys pre-commit hook to v0.3.0 #1139
- chore: use default vale version #1140
- docs: add `user_agent` to Sphinx build #1142
- ci: enabling Linux tests missing #1152
- ci: perform minimal requirements tests #1153

11.24.3 Fixed

- fix: docs link in example #1137
- fix: update backend version message #1145
- fix: Trame issues #1148
- fix: Interactive documentation #1160

11.24.4 Dependencies

- build: bump ansys-tools-path from 0.5.1 to 0.5.2 #1131
- build: bump the grpc-deps group across 1 directory with 3 updates #1156
- build: bump notebook from 7.1.2 to 7.1.3 in the docs-deps group #1157
- build: bump beartype from 0.18.2 to 0.18.5 #1158

11.24.5 Miscellaneous

- docs: add example on exporting designs #1149
- docs: fix link in *CHANGELOG.md* #1154
- chore: pre-commit automatic update #1159

11.25 0.5.0 - 2024-04-17

11.25.1 Added

- feat: inserting document into existing design #930
- feat: add changelog action #1023
- feat: create a sphere body on the backend #1035
- feat: mirror a body #1055
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- feat: avoid error if folder exists #1125

11.25.2 Changed

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- chore: update pre-commit versions #1094
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11.25.3 Fixed

- feat: re-enable open file on Linux #817
- fix: adapt export and download tests to new hoops #1057
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- fix: unit tests failing after dms update #1087
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- fix: improper types being passed for Face and Edge ctor. #1096
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11.25.4 Dependencies

- build: bump the docs-deps group with 2 updates #1062, #1093, #1105
- build: bump ansys-api-geometry from 0.3.13 to 0.4.0 #1066
- build: bump the docs-deps group with 1 update #1080
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- build: bump ansys-api-geometry from 0.4.0 to 0.4.1 #1092
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11.25.5 Miscellaneous

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- docs: add examples on new methods #1089
- chore: pre-commit automatic update #1116

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