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PyAnsys Geometry

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PyAnsys Geometry

PyAnsys Geometry is a Python client library for the Ansys Geometry service. You are looking at the documentation for version 0.8.dev0.

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.

CHAPTER

ONE

INSTALLATION

You can use pip to install PyAnsys Geometry.

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

CHAPTER

TWO

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

CHAPTER

THREE

COMPATIBILITY WITH ANSYS RELEASES

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

CHAPTER

FOUR

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*.

CHAPTER

FIVE

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

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 XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX > 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
>>> 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.

```python
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 2023 R2 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

```python
from ansys.geometry.core import launch_modeler_with_discovery

modeler = launch_modeler_with_discovery()
```

SpaceClaim

```python
from ansys.geometry.core import launch_modeler_with_spaceclaim

modeler = launch_modeler_with_spaceclaim()
```

Geometry service

```python
from ansys.geometry.core import launch_modeler_with_geometry_service

modeler = launch_modeler_with_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 2023 R2 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:

```python
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

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 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 3.12. 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-v0.8.dev0-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)
```

(continues on next page)

(continued from previous page)

```
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*.

Go to Getting started

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 automated installer, the Ansys Geometry service is installed by default.

However, when using the unified 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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CHAPTER SIX

USER GUIDE

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"
)
sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&quot;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))
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])
```

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```
# 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-...
```

CHAPTER

SEVEN

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

connection	PyAnsys Geometry connection subpackage.
designer	PyAnsys Geometry designer subpackage.
materials	PyAnsys Geometry materials subpackage.
math	PyAnsys Geometry math subpackage.
misc	Provides the PyAnsys Geometry miscellaneous subpackage.
plotting	Provides the PyAnsys Geometry plotting subpackage.
shapes	Provides the PyAnsys Geometry `geometry` subpackage.
sketch	PyAnsys Geometry sketch subpackage.
tools	PyAnsys Geometry tools subpackage.

Submodules

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

Attributes

__version__	PyAnsys Geometry version.

Constants

USE_SERVICE_COLORS	Global constant for checking whether to use service colors for plotting
DISABLE_MULTIPLE_DESIGN_CHECK	Global constant for disabling the `ensure_design_is_active` check.
DOCUMENTATION_BUILD	Global flag for the documentation to use the proper PyVista Jupyter backend.

The `connection` package

Summary

Submodules

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

The `backend.py` module

Summary

Enums

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

BackendType

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

Bases: `enum.Enum`

Provides an enum holding the available backend types.

Overview

Attributes

DISCOVERY
SPACECLAIM
WINDOWS_SERVICE
LINUX_SERVICE

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`

ApiVersions

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

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

Static methods

parse_input	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**

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

GrpcClient	Wraps the gRPC connection for the Geometry service.

Functions

wait_until_healthy	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: ansys.platform.instancemanagement.Instance | None = None*, *docker_instance:* ansys.geometry.core.connection.docker_instance.LocalDockerInst | *None = None*, *product_instance:* ansys.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*)

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 remote 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 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] 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

close	Close the channel.
target	Get the target of the channel.
get_name	Get the target name of the connection.

Properties

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

Attributes

grpc_backend_response

Special methods

__repr__	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

This method will return `False` if the backend type is `Discovery` or `Linux Service`. Otherwise, it will return `True`.

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.

Attribute detail

GrpcClient.**grpc_backend_response**

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. An attempt is made every 100 milliseconds until the timeout is exceeded.

Raises

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

The `conversions.py` module

Summary

Functions

unit_vector_to_grpc_direction	Convert a `UnitVector3D` class to a unit vector gRPC message.
frame_to_grpc_frame	Convert a `Frame` class to a frame gRPC message.
plane_to_grpc_plane	Convert a `Plane` class to a plane gRPC message.
sketch_shapes_to_grpc_geometries	Convert lists of `SketchEdge` and `SketchFace` to a gRPC message.
sketch_edges_to_grpc_geometries	Convert a list of `SketchEdge` to a gRPC message.
sketch_arc_to_grpc_arc	Convert an `Arc` class to an arc gRPC message.
sketch_ellipse_to_grpc_ellipse	Convert a `SketchEllipse` class to an ellipse gRPC message.
sketch_circle_to_grpc_circle	Convert a `SketchCircle` class to a circle gRPC message.
point3d_to_grpc_point	Convert a `Point3D` class to a point gRPC message.
point2d_to_grpc_point	Convert a `Point2D` class to a point gRPC message.
sketch_polygon_to_grpc_polygon	Convert a `Polygon` class to a polygon gRPC message.
sketch_segment_to_grpc_line	Convert a `Segment` class to a line gRPC message.
tess_to_pd	Convert an `ansys.api.geometry.Tessellation` to `pyvista.PolyData`.
grpc_matrix_to_matrix	Convert an `ansys.api.geometry.Matrix` to a `Matrix44`.
grpc_frame_to_frame	Convert a frame gRPC message to a `Frame` class.
grpc_surface_to_surface	Convert a surface gRPC message to a `Surface` class.
grpc_curve_to_curve	Convert a curve gRPC message to a `Curve`.
curve_to_grpc_curve	Convert a `Curve` object to a curve gRPC message.
trimmed_curve_to_grpc_trimmed_cur	Convert a `TrimmedCurve` to a trimmed curve gRPC message.

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.`**`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.

The defaults.py module

Summary

Constants

DEFAULT_HOST	Default for the HOST name.
DEFAULT_PORT	Default for the HOST port.
MAX_MESSAGE_LENGTH	Default for the gRPC maximum message length.
GEOMETRY_SERVICE_DOCKER_IMAGE	Default for the Geometry service Docker image location.
DEFAULT_PIM_CONFIG	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

LocalDockerInstance	Instantiates a Geometry service as a local Docker container.

Enums

GeometryContainers	Provides an enum holding the available Geometry services.

Functions

get_geometry_container_type	Provide back the GeometryContainers 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

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

Attributes

__DOCKER_CLIENT__	Docker client class variable.

Static methods

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

Import detail

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

Property detail

property LocalDockerInstance.**container**: docker.models.containers.Container

Docker container object that hosts the deployed Geometry service.

property LocalDockerInstance.**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

LocalDockerInstance.**__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 LocalDockerInstance.**docker_client**() → docker.client.DockerClient

Get the initialized __DOCKER_CLIENT__ object.

Returns

DockerClient
Initialized Docker client.

Notes

The LocalDockerInstance class performs a lazy initialization of the __DOCKER_CLIENT__ class variable.

static LocalDockerInstance.**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*, ***kwds*)

Bases: enum.Enum

Provides an enum holding the available Geometry services.

Overview

Attributes

WINDOWS_LATEST
LINUX_LATEST
WINDOWS_LATEST_UNSTABLE
LINUX_LATEST_UNSTABLE
WINDOWS_24_1
LINUX_24_1
WINDOWS_24_2
LINUX_24_2

Import detail

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

Attribute detail

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

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

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

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

`GeometryContainers.WINDOWS_24_1 = (4, 'windows', 'windows-24.1')`

`GeometryContainers.LINUX_24_1 = (5, 'linux', 'linux-24.1')`

`GeometryContainers.WINDOWS_24_2 = (6, 'windows', 'windows-24.2')`

`GeometryContainers.LINUX_24_2 = (7, 'linux', 'linux-24.2')`

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

launch_modeler	Start the Modeler interface for PyAnsys Geometry.
launch_remote_modeler	Start the Geometry service remotely using the PIM API.
launch_docker_modeler	Start the Geometry service locally using Docker.
launch_modeler_with_discovery_and_pimlight	Start Ansys Discovery remotely using the PIM API.
launch_modeler_with_geometry_service_and_p	Start the Geometry service remotely using the PIM API.
launch_modeler_with_spaceclaim_and_pimligh	Start Ansys SpaceClaim remotely using the PIM API.
launch_modeler_with_geometry_service	Start the Geometry service locally using the ProductInstance class.
launch_modeler_with_discovery	Start Ansys Discovery locally using the ProductInstance class.
launch_modeler_with_spaceclaim	Start Ansys SpaceClaim 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 "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, "232". 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, "232". 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, "232". 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, "232". 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 v23.2.1 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

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 23.2 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.

Note

Support for Ansys Discovery is restricted to Ansys 24.1 onward.

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 v23.2.1 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

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 23.2 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 23.2 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 = 232,
    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 v23.2.1 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:

- `232` : Ansys 23R2 SP1

- `241` : Ansys 24R1

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 23.2 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 23.2 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 = 232,
    host="10.171.22.44",
    port=5001,
    api_version= 231,
    timeout=300,
    server_log_level=0)
```

The `product_instance.py` module

Summary

Classes

ProductInstance	`ProductInstance` class.

Functions

prepare_and_start_backend	Start the requested service locally using the `ProductInstance` class.
get_available_port	Return an available port to be used.

Constants

WINDOWS_GEOMETRY_SERVICE_FOLDER	Default Geometry Service's folder name into the unified installer.
DISCOVERY_FOLDER	Default Discovery's folder name into the unified installer.
SPACECLAIM_FOLDER	Default SpaceClaim's folder name into the unified installer.
ADDINS_SUBFOLDER	Default global Addins's folder name into the unified installer.
BACKEND_SUBFOLDER	Default backend's folder name into the ADDINS_SUBFOLDER folder.
MANIFEST_FILENAME	Default backend's add-in filename.
GEOMETRY_SERVICE_EXE	The Windows Geometry Service's filename.
DISCOVERY_EXE	The Ansys Discovery's filename.
SPACECLAIM_EXE	The Ansys SpaceClaim's filename.
BACKEND_LOG_LEVEL_VARIABLE	The backend's log level environment variable for local start.
BACKEND_TRACE_VARIABLE	The backend's enable trace environment variable for local start.
BACKEND_HOST_VARIABLE	The backend's ip address environment variable for local start.
BACKEND_PORT_VARIABLE	The backend's port number environment variable for local start.
BACKEND_LOGS_FOLDER_VARIABLE	The backend's logs folder path to be used.
BACKEND_API_VERSION_VARIABLE	The backend's api version environment variable for local start.
BACKEND_SPACECLAIM_OPTIONS	The additional argument for local Ansys Discovery start.
BACKEND_ADDIN_MANIFEST_ARGUMENT	The argument to specify the backend's add-in manifest file's path.
BACKEND_SPACECLAIM_HIDDEN	The argument to hide SpaceClaim's UI on the backend.
BACKEND_SPACECLAIM_HIDDEN_ENVVAR_KEY	SpaceClaim hidden backend's environment variable key.
BACKEND_SPACECLAIM_HIDDEN_ENVVAR_VALU	SpaceClaim hidden backend's environment variable value.
BACKEND_DISCOVERY_HIDDEN	The argument to hide Discovery's UI on the backend.
BACKEND_SPLASH_OFF	The argument to specify the backend's add-in manifest file's path.

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

close	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*, *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 v23.2.1 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.

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 23.2 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.

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.

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.

The `validate.py` module

Summary

Functions

validate	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

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

The beam.py module

Summary

Classes

BeamProfile	Represents a single beam profile organized within the design assembly.
BeamCircularProfile	Represents a single circular beam profile.
Beam	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

id	ID of the beam profile.
name	Name of the beam profile.

Import detail

```python
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: ansys.geometry.core.misc.measurements.Distance*, *center: ansys.geometry.core.math.point.Point3D*, *direction_x: ansys.geometry.core.math.vector.UnitVector3D*, *direction_y: ansys.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

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

Special methods

__repr__	Represent the BeamCircularProfile as a string.

Import detail

```python
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

id	Service-defined ID of the beam.
start	Start of the beam line segment.
end	End of the beam line segment.
profile	Beam profile of the beam line segment.
parent_component	Component node that the beam is under.
is_alive	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

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

Classes

MasterBody	Represents solids and surfaces organized within the design assembly.
Body	Represents solids and surfaces organized within the design assembly.

Enums

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

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

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

Methods

intersect	Intersect two (or more) bodies.
subtract	Subtract two (or more) bodies.
unite	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.**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.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.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)
>>> 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
```

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```
Z Bounds:      -5.000e-01, 4.500e+00
N Arrays:      0
```

abstract IBody.**plot**(*merge: bool = False*, *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: 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.

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

****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: ansys.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

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

Methods

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

Properties

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

Special methods

__repr__	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.**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.

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.`**`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_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
```

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```
Y Bounds:     -1.000e+00, 0.000e+00
Z Bounds:     -5.000e-01, 4.500e+00
N Arrays:     0
```

abstract MasterBody.**plot**(*merge: bool = False*, *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: 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.

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

****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

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

Properties

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

Special methods

__repr__	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.**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.

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.**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_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.**plot**(*merge: bool = False*, *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: `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.

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

****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*, ***kwds*)

Bases: enum.Enum

Provides values for mid-surface offsets supported.

Overview

Attributes

MIDDLE
TOP
BOTTOM
VARIABLE
CUSTOM

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*, ***kwds*)

Bases: enum.Enum

Provides values for collision types between bodies.

Overview

Attributes

NONE
TOUCH
INTERSECT
CONTAINED
CONTAINEDTOUCH

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*, ***kwds*)

Bases: enum.Enum

Provides values for fill styles supported.

Overview

Attributes

DEFAULT
OPAQUE
TRANSPARENT

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.

The component.py module

Summary

Classes

Component	Provides for creating and managing a component.

Enums

SharedTopologyType	Shared topologies available.
ExtrusionDirection	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 conponent.

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

get_world_transform	Get the full transformation matrix of the component in world space.
modify_placement	Apply a translation and/or rotation to the placement matrix.
reset_placement	Reset a component's placement matrix to an identity matrix.
add_component	Add a new component under this component within the design assembly.
set_shared_topology	Set the shared topology to apply to the component.
extrude_sketch	Create a solid body by extruding the sketch profile a distance.
sweep_sketch	Create a body by sweeping a planar profile along a path.
sweep_chain	Create a body by sweeping a chain of curves along a path.
revolve_sketch	Create a solid body by revolving a sketch profile around an axis.
extrude_face	Extrude the face profile by a given distance to create a solid body.
create_sphere	Create a sphere body defined by the center point and the radius.
create_body_from_loft_profile	Create a lofted body from a collection of trimmed curves.
create_surface	Create a surface body with a sketch profile.
create_surface_from_face	Create a surface body based on a face.
create_coordinate_system	Create a coordinate system.
translate_bodies	Translate the bodies in a specified direction by a distance.
create_beams	Create beams under the component.
create_beam	Create a beam under the component.
delete_component	Delete a component (itself or its children).
delete_body	Delete a body belonging to this component (or its children).
add_design_point	Create a single design point.
add_design_points	Create a list of design points.
delete_beam	Delete an existing beam belonging to this component's scope.
search_component	Search nested components recursively for a component.
search_body	Search bodies in the component's scope.
search_beam	Search beams in the component's scope.
tessellate	Tessellate the component.
plot	Plot the component.
tree_print	Print the component in tree format.

Properties

instance_name	Name of the component instance.
id	ID of the component.
name	Name of the component.
components	List of `Component` objects inside of the component.
bodies	List of `Body` objects inside of the component.
beams	List of `Beam` objects inside of the component.
design_points	List of `DesignPoint` objects inside of the component.
coordinate_systems	List of `CoordinateSystem` objects inside of the component.
parent_component	Parent of the component.
is_alive	Whether the component is still alive on the server side.
shared_topology	Shared topology type of the component (if any).

Special methods

__repr__	Represent the Component as a string.

Import detail

```
from ansys.geometry.core.designer.component import Component
```

Property detail

property Component.**instance_name: str**

Name of the component instance.

property Component.**id: str**

ID of the component.

property Component.**name: str**

Name of the component.

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_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*) → *ansys.geometry.core.designer.body.Body*

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.

Returns

Body
Extruded body from the given sketch.

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_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**(*merge_component: bool = False*, *merge_bodies: bool = False*) → pyvista.PolyData | pyvista.MultiBlock

Tessellate the component.

Parameters

merge_component
[bool, default: False] Whether to merge this 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 merged into a single dataset without separating faces.

Returns

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

Examples

Create two stacked bodies and return the tessellation as two merged bodies:

```
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core import Modeler
>>> from ansys.geometry.core.math import Point2D, Point3D, Plane
>>> from ansys.geometry.core.misc import UNITS
>>> modeler = Modeler("10.54.0.72", "50051")
>>> sketch_1 = Sketch()
>>> box = sketch_1.box(
>>>    Point2D([10, 10], UNITS.m), Quantity(10, UNITS.m), Quantity(5, UNITS.m))
>>> sketch_1.circle(Point2D([0, 0], UNITS.m), Quantity(25, UNITS.m))
>>> design = modeler.create_design("MyDesign")
>>> comp = design.add_component("MyComponent")
>>> distance = Quantity(10, UNITS.m)
>>> body = comp.extrude_sketch("Body", sketch=sketch_1, distance=distance)
>>> sketch_2 = Sketch(Plane([0, 0, 10]))
>>> box = sketch_2.box(
>>>    Point2D([10, 10], UNITS.m), Quantity(10, UNITS.m), Quantity(5, UNITS.m))
>>> circle = sketch_2.circle(Point2D([0, 0], UNITS.m), Quantity(25, UNITS.m))
>>> body = comp.extrude_sketch("Body", sketch=sketch_2, distance=distance)
>>> dataset = comp.tessellate(merge_bodies=True)
>>> dataset
MultiBlock (0x7ff6bcb511e0)
  N Blocks:     2
  X Bounds:     -25.000, 25.000
  Y Bounds:     -24.991, 24.991
  Z Bounds:     0.000, 20.000
```

Component.**plot**(*merge_component: bool = False*, *merge_bodies: bool = False*, *screenshot: str | None = None*, *use_trame: bool | None = None*, *use_service_colors: bool | None = None*, ***plotting_options: dict | None*) → None

Plot the component.

Parameters

merge_component
[bool, default: `False`] Whether to merge the component into a single dataset. When `True`, all the individual bodies are effectively merged 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 merged into a single dataset without separating faces.

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.

****plotting_options**
[dict, default: None] Keyword arguments for plotting. For allowable keyword arguments, see the

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*, ***kwds*)

Bases: enum.Enum

Shared topologies available.

Overview

Attributes

SHARETYPE_NONE
SHARETYPE_SHARE
SHARETYPE_MERGE
SHARETYPE_GROUPS

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*, ***kwds*)

Bases: enum.Enum

Enum for extrusion direction definition.

Overview

Constructors

from_string	Convert a string to an ExtrusionDirection enum.

Attributes

POSITIVE
NEGATIVE

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

CoordinateSystem	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.Com *grpc_client:* ansys.geometry.core.connection.client.GrpcCl *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

id	ID of the coordinate system.
name	Name of the coordinate system.
frame	Frame of the coordinate system.
parent_component	Parent component of the coordinate system.
is_alive	Flag indicating if coordinate system is still alive on the server.

Attributes

new_coordinate_system

Special methods

__repr__	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.

Attribute detail

CoordinateSystem.**new_coordinate_system**

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

Design	Provides for organizing geometry assemblies.

Enums

DesignFileFormat	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

close	Close the design.
add_material	Add a material to the design.
save	Save a design to disk on the active Geometry server instance.
download	Export and download the design from the server.
export_to_scdocx	Export the design to an scdocx file.
export_to_parasolid_text	Export the design to a Parasolid text file.
export_to_parasolid_bin	Export the design to a Parasolid binary file.
export_to_fmd	Export the design to an FMD file.
export_to_step	Export the design to a STEP file.
export_to_iges	Export the design to an IGES file.
export_to_pmdb	Export the design to a PMDB file.
create_named_selection	Create a named selection on the active Geometry server instance.
delete_named_selection	Delete a named selection on the active Geometry server instance.
delete_component	Delete a component (itself or its children).
set_shared_topology	Set the shared topology to apply to the component.
add_beam_circular_profile	Add a new beam circular profile under the design for creating beams.
add_midsurface_thickness	Add a mid-surface thickness to a list of bodies.
add_midsurface_offset	Add a mid-surface offset type to a list of bodies.
delete_beam_profile	Remove a beam profile on the active geometry server instance.
insert_file	Insert a file into the design.

Properties

design_id	The design's object unique id.
materials	List of materials available for the design.
named_selections	List of named selections available for the design.
beam_profiles	List of beam profile available for the design.
is_active	Whether the design is currently active.
is_closed	Whether the design is closed.

Special methods

__repr__	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.**is_active**: bool

Whether the design is currently active.

property Design.**is_closed**: bool

Whether the design is closed.

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.SCDOCX`] Format for the file to save to.

`Design.export_to_scdocx`(*location: pathlib.Path | str | None = None*) → pathlib.Path

Export the design to an scdocx 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.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*) → *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.

Returns

Component
The newly inserted component.

Design.**__repr__**() → str

Represent the Design as a string.

DesignFileFormat

class ansys.geometry.core.designer.design.**DesignFileFormat**(**args*, ***kwds*)

Bases: enum.Enum

Provides supported file formats that can be downloaded for designs.

Overview

Attributes

SCDOCX
PARASOLID_TEXT
PARASOLID_BIN
FMD
STEP
IGES
PMDB
INVALID

Import detail

```
from ansys.geometry.core.designer.design import DesignFileFormat
```

Attribute detail

DesignFileFormat.**SCDOCX = ('SCDOCX', None)**

DesignFileFormat.**PARASOLID_TEXT**

DesignFileFormat.**PARASOLID_BIN**

DesignFileFormat.**FMD**

DesignFileFormat.**STEP**

DesignFileFormat.**IGES**

DesignFileFormat.**PMDB**

DesignFileFormat.**INVALID = ('INVALID', None)**

Description

Provides for managing designs.

The `designpoint.py` module

Summary

Classes

DesignPoint	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

id	ID of the design point.
name	Name of the design point.
value	Value of the design point.
parent_component	Component node that the design point is under.

Special methods

__repr__	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

Edge	Represents a single edge of a body within the design assembly.

Enums

CurveType	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

id	ID of the edge.
is_reversed	Flag indicating if the edge is reversed.
shape	Underlying trimmed curve of the edge.
length	Calculated length of the edge.
curve_type	Curve type of the edge.
faces	Faces that contain the edge.
start	Start point of the edge.
end	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.**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*, ***kwds*)

Bases: enum.Enum

Provides values for the curve types supported.

Overview

Attributes

CURVETYPE_UNKNOWN
CURVETYPE_LINE
CURVETYPE_CIRCLE
CURVETYPE_ELLIPSE
CURVETYPE_NURBS
CURVETYPE_PROCEDURAL

Import detail

```python
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

FaceLoop	Provides an internal class holding the face loops defined.
Face	Represents a single face of a body within the design assembly.

Enums

SurfaceType	Provides values for the surface types supported.
FaceLoopType	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

type	Type of the loop.
length	Length of the loop.
min_bbox	Minimum point of the bounding box containing the loop.
max_bbox	Maximum point of the bounding box containing the loop.
edges	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

normal	Get the normal direction to the face at certain UV coordinates.
face_normal	Get the normal direction to the face at certain UV coordinates.
point	Get a point of the face evaluated at certain UV coordinates.
face_point	Get a point of the face evaluated at certain UV coordinates.
create_isoparametric_curves	Create isoparametric curves at the given proportional parameter.

Properties

id	Face ID.
is_reversed	Flag indicating if the face is reversed.
body	Body that the face belongs to.
shape	Underlying trimmed surface of the face.
surface_type	Surface type of the face.
area	Calculated area of the face.
edges	List of all edges of the face.
loops	List of all loops of 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.

Method detail

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 isoparametic 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.

SurfaceType

class ansys.geometry.core.designer.face.**SurfaceType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for the surface types supported.

Overview

Attributes

SURFACETYPE_UNKNOWN
SURFACETYPE_PLANE
SURFACETYPE_CYLINDER
SURFACETYPE_CONE
SURFACETYPE_TORUS
SURFACETYPE_SPHERE
SURFACETYPE_NURBS
SURFACETYPE_PROCEDURAL

Import detail

```
from ansys.geometry.core.designer.face import SurfaceType
```

Attribute detail

SurfaceType.**SURFACETYPE_UNKNOWN = 0**

SurfaceType.**SURFACETYPE_PLANE = 1**

SurfaceType.**SURFACETYPE_CYLINDER = 2**

SurfaceType.**SURFACETYPE_CONE = 3**

SurfaceType.**SURFACETYPE_TORUS = 4**

SurfaceType.**SURFACETYPE_SPHERE = 5**

SurfaceType.**SURFACETYPE_NURBS = 6**

SurfaceType.**SURFACETYPE_PROCEDURAL = 7**

FaceLoopType

class ansys.geometry.core.designer.face.**FaceLoopType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for the face loop types supported.

Overview

Attributes

INNER_LOOP	
OUTER_LOOP	

Import detail

```python
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 part.py module

Summary

Classes

Part	Represents a part master.
MasterComponent	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

id	ID of the part.
name	Name of the part.
components	`MasterComponent` children that the part contains.
bodies	`MasterBody` children that the part contains.

Special methods

__repr__	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

id	ID of the transformed part.
name	Name of the transformed part.
occurrences	List of all occurrences of the component.
part	Master part of the transformed part.
transform	4x4 transformation matrix from the master part.

Special methods

__repr__	Represent the master component as a string.

Import detail

```python
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

NamedSelection	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.DesignPoi | 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

id	ID of the named selection.
name	Name of the named selection.

Attributes

ids
named_selection_request
new_named_selection

Import detail

```python
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.

Attribute detail

NamedSelection.**ids**

NamedSelection.**named_selection_request**

NamedSelection.**new_named_selection**

Description

Module for creating a named selection.

Description

PyAnsys Geometry designer subpackage.

The `materials` package

Summary

Submodules

material	Provides the data structure for material and material properties.
property	Provides the `MaterialProperty` class.

The `material.py` module

Summary

Classes

Material	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.Mate
| *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

add_property	Add a material property to the `Material` class.

Properties

properties	Dictionary of the material property type and material properties.
name	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

MaterialProperty	Provides the data structure for a material property.

Enums

MaterialPropertyType	Enum holding the possible values for `MaterialProperty` 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

type	Material property ID.
name	Material property name.
quantity	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*, ***kwds*)

Bases: enum.Enum

Enum holding the possible values for MaterialProperty objects.

Overview

Attributes

DENSITY
ELASTIC_MODULUS
POISSON_RATIO
SHEAR_MODULUS
SPECIFIC_HEAT
TENSILE_STRENGTH
THERMAL_CONDUCTIVITY

Static methods

from_id	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

bbox	Provides for managing a bounding box.
constants	Provides mathematical constants.
frame	Provides for managing a frame.
matrix	Provides matrix primitive representations.
misc	Provides auxiliary math functions for PyAnsys Geometry.
plane	Provides primitive representation of a 2D plane in 3D space.
point	Provides geometry primitive representation for 2D and 3D points.
vector	Provides for creating and managing 2D and 3D vectors.

The `bbox.py` module

Summary

Classes

BoundingBox2D	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

add_point	Extend the ranges of the bounding box to include a point.
add_point_components	Extend the ranges of the bounding box to include the X and Y values.
add_points	Extend the ranges of the bounding box to include given points.
contains_point	Evaluate whether a point lies within the X and Y range bounds.
contains_point_components	Check if point components are within the X and Y range bounds.

Properties

x_min	Minimum value of X-dimensional bounds.
x_max	Maximum value of the X-dimensional bounds.
y_min	Minimum value of Y-dimensional bounds.
y_max	Maximum value of Y-dimensional bounds.

Special methods

__eq__	Equals operator for the `BoundingBox2D` class.
__ne__	Not equals operator for the `BoundingBox2D` class.

Import detail

```python
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.

Description

Provides for managing a bounding box.

The constants.py module

Summary

Constants

DEFAULT_POINT3D	Default value for a 3D point.
DEFAULT_POINT2D	Default value for a 2D point.
IDENTITY_MATRIX33	Identity for a Matrix33 object.
IDENTITY_MATRIX44	Identity for a Matrix44 object.
UNITVECTOR3D_X	Default 3D unit vector in the Cartesian traditional X direction.
UNITVECTOR3D_Y	Default 3D unit vector in the Cartesian traditional Y direction.
UNITVECTOR3D_Z	Default 3D unit vector in the Cartesian traditional Z direction.
UNITVECTOR2D_X	Default 2D unit vector in the Cartesian traditional X direction.
UNITVECTOR2D_Y	Default 2D unit vector in the Cartesian traditional Y direction.
ZERO_VECTOR3D	Zero-valued Vector3D object.
ZERO_VECTOR2D	Zero-valued Vector2D object.
ZERO_POINT3D	Zero-valued Point3D object.
ZERO_POINT2D	Zero-valued Point2D 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

Frame	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

transform_point2d_local_to_global	Transform a 2D point to a global 3D point.

Properties

origin	Origin of the frame.
direction_x	X-axis direction of the frame.
direction_y	Y-axis direction of the frame.
direction_z	Z-axis direction of the frame.
global_to_local_rotation	Global to local space transformation matrix.
local_to_global_rotation	Local to global space transformation matrix.
transformation_matrix	Full 4x4 transformation matrix.

Special methods

`__eq__`	Equals operator for the `Frame` class.
`__ne__`	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

Matrix	Provides matrix representation.
Matrix33	Provides 3x3 matrix representation.
Matrix44	Provides 4x4 matrix representation.

Constants

DEFAULT_MATRIX33	Default value of the 3x3 identity matrix for the `Matrix33` class.
DEFAULT_MATRIX44	Default value of the 4x4 identity matrix for the `Matrix44` 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

determinant	Get the determinant of the matrix.
inverse	Provide the inverse of the matrix.

Special methods

__mul__	Get the multiplication of the matrix.
__eq__	Equals operator for the Matrix class.
__ne__	Not equals operator for the Matrix 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.

Import detail

```
from ansys.geometry.core.math.matrix import Matrix44
```

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

get_two_circle_intersections	Get the intersection points of two circles.

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/

The `plane.py` module

Summary

Classes

Plane	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

is_point_contained	Check if a 3D point is contained in the plane.

Properties

normal	Calculate the normal vector of the plane.

Special methods

__eq__	Equals operator for the `Plane` class.
__ne__	Not equals operator for the `Plane` 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

Point2D	Provides geometry primitive representation for a 2D point.
Point3D	Provides geometry primitive representation for a 3D point.

Constants

DEFAULT_POINT2D_VALUES	Default values for a 2D point.
DEFAULT_POINT3D_VALUES	Default values for a 3D point.
BASE_UNIT_LENGTH	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

base_unit	Get the base unit of the object.

Properties

x	X plane component value.
y	Y plane component value.

Attributes

unit	Unit of the object.
flat	

Special methods

__eq__	Equals operator for the `Point2D` class.
__ne__	Not equals operator for the `Point2D` class.
__add__	Add operation for the `Point2D` class.
__sub__	Subtraction operation for the `Point2D` 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.unit`
Unit of the object.

`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.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

base_unit	Get the base unit of the object.
transform	Transform the 3D point with a transformation matrix.

Properties

x	X plane component value.
y	Y plane component value.
z	Z plane component value.

Attributes

unit	Unit of the object.
flat	

Special methods

__eq__	Equals operator for the `Point3D` class.
__ne__	Not equals operator for the `Point3D` class.
__add__	Add operation for the `Point3D` class.
__sub__	Subtraction operation for the `Point3D` 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.**unit**

Unit of the object.

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.**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

Vector3D	Provides for managing and creating a 3D vector.
Vector2D	Provides for creating and managing a 2D vector.
UnitVector3D	Provides for creating and managing a 3D unit vector.
UnitVector2D	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

from_points	Create a 3D vector from two distinct 3D points.

Methods

is_perpendicular_to	Check if this vector and another vector are perpendicular.
is_parallel_to	Check if this vector and another vector are parallel.
is_opposite	Check if this vector and another vector are opposite.
normalize	Return a normalized version of the 3D vector.
transform	Transform the 3D vector3D with a transformation matrix.
get_angle_between	Get the angle between this 3D vector and another 3D vector.
cross	Return the cross product of `Vector3D` objects.

Properties

x	X coordinate of the `Vector3D` class.
y	Y coordinate of the `Vector3D` class.
z	Z coordinate of the `Vector3D` class.
norm	Norm of the vector.
magnitude	Norm of the vector.
is_zero	Check if all components of the 3D vector are zero.

Special methods

__eq__	Equals operator for the `Vector3D` class.
__ne__	Not equals operator for the `Vector3D` class.
__mul__	Overload * operator with dot product.
__mod__	Overload % operator with cross product.
__add__	Addition operation overload for 3D vectors.
__sub__	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 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

from_points	Create a 2D vector from two distinct 2D points.

Methods

cross	Return the cross product of `Vector2D` objects.
is_perpendicular_to	Check if this 2D vector and another 2D vector are perpendicular.
is_parallel_to	Check if this vector and another vector are parallel.
is_opposite	Check if this vector and another vector are opposite.
normalize	Return a normalized version of the 2D vector.
get_angle_between	Get the angle between this 2D vector and another 2D vector.

Properties

x	X coordinate of the 2D vector.
y	Y coordinate of the 2D vector.
norm	Norm of the 2D vector.
magnitude	Norm of the 2D vector.
is_zero	Check if values for all components of the 2D vector are zero.

Special methods

__eq__	Equals operator for the `Vector2D` class.
__ne__	Not equals operator for the `Vector2D` class.
__mul__	Overload * operator with dot product.
__add__	Addition operation overload for 2D vectors.
__sub__	Subtraction operation overload for 2D vectors.
__mod__	Overload % 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

from_points	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

accuracy	Provides for evaluating decimal precision.
auxiliary	Auxiliary functions for the PyAnsys Geometry library.
checks	Provides functions for performing common checks.
measurements	Provides various measurement-related classes.
options	Provides various option classes.
units	Provides for handling units homogeneously throughout PyAnsys Geometry.

The `accuracy.py` module

Summary

Classes

Accuracy	Decimal precision evaluations for math operations.

Constants

LENGTH_ACCURACY	Constant for decimal accuracy in length comparisons.
ANGLE_ACCURACY	Constant for decimal accuracy in angle comparisons.
DOUBLE_ACCURACY	Constant for double accuracy.

`Accuracy`

class `ansys.geometry.core.misc.accuracy.`**`Accuracy`**

Decimal precision evaluations for math operations.

Overview

Static methods

length_accuracy	Return the LENGTH_ACCURACY constant.
angle_accuracy	Return the ANGLE_ACCURACY constant.
double_accuracy	Return the DOUBLE_ACCURACY constant.
length_is_equal	Check if the comparison length is equal to the reference length.
equal_doubles	Compare two double values.
compare_with_tolerance	Compare two doubles given the relative and absolute tolerances.
length_is_greater_than_or_equal	Check if the length is greater than the reference length.
length_is_less_than_or_equal	Check if the length is less than or equal to the reference length.
length_is_zero	Check if the length is within the length accuracy of exact zero.
length_is_negative	Check if the length is below a negative length accuracy.
length_is_positive	Check if the length is above a positive length accuracy.
angle_is_zero	Check if the length is within the angle accuracy of exact zero.
angle_is_negative	Check if the angle is below a negative angle accuracy.
angle_is_positive	Check if the angle is above a positive angle accuracy.
is_within_tolerance	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

get_design_from_component	Get the `Design` of the given `Component` object.
get_design_from_body	Get the `Design` of the given `Body` object.
get_design_from_face	Get the `Design` of the given `Face` object.
get_design_from_edge	Get the `Design` of the given `Edge` object.
get_bodies_from_ids	Find the `Body` objects inside a `Design` from its ids.
get_faces_from_ids	Find the `Face` objects inside a `Design` from its ids.
get_edges_from_ids	Find the `Edge` objects inside a `Design` from its ids.

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_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.

The `checks.py` module

Summary

Functions

ensure_design_is_active	Make sure that the design is active before executing a method.
check_is_float_int	Check if a parameter has a float or integer value.
check_ndarray_is_float_int	Check if a `numpy.ndarray` has float/integer types.
check_ndarray_is_not_none	Check if a `numpy.ndarray` is all `None`.
check_ndarray_is_all_nan	Check if a `numpy.ndarray` is all nan-valued.
check_ndarray_is_non_zero	Check if a `numpy.ndarray` is zero-valued.
check_pint_unit_compatibility	Check if input `pint.Unit` is compatible with the expected input.
check_type_equivalence	Check if an input object is of the same class as an expected object.
check_type	Check if an input object is of the same type as expected types.
check_type_all_elements_in_iterable	Check if all elements in an iterable are of the same type as expected.
min_backend_version	Compare a minimum required version to the current backend version.
deprecated_method	Decorate a method as deprecated.
deprecated_argument	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*)

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.

checks.**deprecated_argument**(*arg: str*, *alternative: str | None = None*, *info: 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.

The `measurements.py` module

Summary

Classes

SingletonMeta	Provides a thread-safe implementation of a singleton design pattern.
DefaultUnitsClass	Provides default units for PyAnsys Geometry.
Measurement	Provides the `PhysicalQuantity` subclass for holding a measurement.
Distance	Provides the `Measurement` subclass for holding a distance.
Angle	Provides the `Measurement` subclass for holding an angle.

Constants

DEFAULT_UNITS	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

__call__	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

LENGTH	Default length unit for PyAnsys Geometry.
ANGLE	Default angle unit for PyAnsys Geometry.
SERVER_LENGTH	Default length unit for gRPC messages.
SERVER_AREA	Default area unit for gRPC messages.
SERVER_VOLUME	Default volume unit for gRPC messages.
SERVER_ANGLE	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

value	Value of the measurement.

Special methods

__eq__	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.**__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.

Overview

Attributes

unit	Unit of the object.

Import detail

```
from ansys.geometry.core.misc.measurements import Distance
```

Attribute detail

Distance.**unit**
Unit of the object.

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.

Overview

Attributes

unit	Unit of the object.

Import detail

```
from ansys.geometry.core.misc.measurements import Angle
```

Attribute detail

Angle.**unit**

Unit of the object.

Description

Provides various measurement-related classes.

Module detail

measurements.**DEFAULT_UNITS**

PyAnsys Geometry default units object.

The `options.py` module

Summary

Classes

ImportOptions	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

to_dict	Provide the dictionary representation of the ImportOptions class.

Attributes

cleanup_bodies
import_coordinate_systems
import_curves
import_hidden_components_and_geometry
import_names
import_planes
import_points

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

PhysicalQuantity	Provides the base class for handling units throughout PyAnsys Geometry.

Constants

UNITS	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

unit	Unit of the object.
base_unit	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 `plotting` package

Summary

Subpackages

widgets	Submodule providing widgets for the PyAnsys Geometry plotter.

Submodules

plotter	Provides plotting for various PyAnsys Geometry objects.

The `widgets` package

Summary

Submodules

show_design_point	Provides the ruler widget for the PyAnsys Geometry plotter.

The `show_design_point.py` module

Summary

Classes

ShowDesignPoints	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.b*

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

callback	Remove or add the DesignPoint actors upon click.
update	Define the configuration and representation of the button widget.

Attributes

plotter_helper	

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

GeometryPlotter	Plotter for PyAnsys Geometry objects.

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

add_frame	Plot a frame in the scene.
add_plane	Plot a plane in the scene.
add_sketch	Plot a sketch in the scene.
add_body_edges	Add the outer edges of a body to the plot.
add_body	Add a body to the scene.
add_component	Add a component to the scene.
add_component_by_body	Add a component on a per body basis.
add_sketch_polydata	Add sketches to the scene from PyVista polydata.
add_design_point	Add a DesignPoint object to the plotter.
plot_iter	Add a list of any type of object to the scene.
plot	Add a custom mesh to the plotter.
show	Show the plotter.

Properties

use_service_colors	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_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, ***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

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.

Description

Provides plotting for various PyAnsys Geometry objects.

Description

Provides the PyAnsys Geometry plotting subpackage.

The `shapes` package

Summary

Subpackages

curves	Provides the PyAnsys Geometry `curves` subpackage.
surfaces	Provides the PyAnsys Geometry `surface` subpackage.

Submodules

box_uv	Provides the BoxUV class.
parameterization	Provides the parametrization-related classes.

The curves package

Summary

Submodules

circle	Provides for creating and managing a circle.
curve	Provides the Curve class.
curve_evaluation	Provides for creating and managing a curve.
ellipse	Provides for creating and managing an ellipse.
line	Provides for creating and managing a line.
trimmed_curve	Trimmed curve class.

The circle.py module

Summary

Classes

Circle	Provides 3D circle representation.
CircleEvaluation	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* | ansys.geometry.core.misc.measurements.Distance | *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.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

contains_param	Check a parameter is within the parametric range of the curve.
contains_point	Check a point is contained by the curve.

Methods

evaluate	Evaluate the circle at a given parameter.
transformed_copy	Create a transformed copy of the circle from a transformation matrix.
mirrored_copy	Create a mirrored copy of the circle along the y-axis.
project_point	Project a point onto the circle and evauate the circle.
is_coincident_circle	Determine if the circle is coincident with another.
parameterization	Get the parametrization of the circle.

Properties

origin	Origin of the circle.
radius	Radius of the circle.
diameter	Diameter of the circle.
perimeter	Perimeter of the circle.
area	Area of the circle.
dir_x	X-direction of the circle.
dir_y	Y-direction of the circle.
dir_z	Z-direction of the circle.

Special methods

`__eq__`	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 evauate 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 2*pi.

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

circle	Circle being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
tangent	Tangent of the evaluation.
normal	Normal to the circle.
first_derivative	First derivative of the evaluation.
second_derivative	Second derivative of the evaluation.
curvature	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

Curve	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

parameterization	Parameterize the curve.
contains_param	Check a parameter is within the parametric range of the curve.
contains_point	Check a point is contained by the curve.
transformed_copy	Create a transformed copy of the curve.
__eq__	Determine if two curves are equal.
evaluate	Evaluate the curve at the given parameter.
project_point	Project a point to the curve.

Methods

trim	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

CurveEvaluation	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

is_set	Determine if the parameter for the evaluation has been set.

Properties

parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
first_derivative	First derivative of the evaluation.
second_derivative	Second derivative of the evaluation.
curvature	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

Ellipse	Provides 3D ellipse representation.
EllipseEvaluation	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* | 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*, *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.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

contains_param	Check a parameter is within the parametric range of the curve.
contains_point	Check a point is contained by the curve.

Methods

mirrored_copy	Create a mirrored copy of the ellipse along the y-axis.
evaluate	Evaluate the ellipse at the given parameter.
project_point	Project a point onto the ellipse and evaluate the ellipse.
is_coincident_ellipse	Determine if this ellipse is coincident with another.
transformed_copy	Create a transformed copy of the ellipse from a transformation matrix.
parameterization	Get the parametrization of the ellipse.

Properties

origin	Origin of the ellipse.
major_radius	Major radius of the ellipse.
minor_radius	Minor radius of the ellipse.
dir_x	X-direction of the ellipse.
dir_y	Y-direction of the ellipse.
dir_z	Z-direction of the ellipse.
eccentricity	Eccentricity of the ellipse.
linear_eccentricity	Linear eccentricity of the ellipse.
semi_latus_rectum	Semi-latus rectum of the ellipse.
perimeter	Perimeter of the ellipse.
area	Area of the ellipse.

Special methods

__eq__	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 2*pi.

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: ansys.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

ellipse	Ellipse being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
tangent	Tangent of the evaluation.
normal	Normal of the evaluation.
first_derivative	Girst derivative of the evaluation.
second_derivative	Second derivative of the evaluation.
curvature	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*

Girst 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

Line	Provides 3D line representation.
LineEvaluation	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

contains_param	Check a parameter is within the parametric range of the curve.
contains_point	Check a point is contained by the curve.

Methods

evaluate	Evaluate the line at a given parameter.
transformed_copy	Create a transformed copy of the line from a transformation matrix.
project_point	Project a point onto the line and evaluate the line.
is_coincident_line	Determine if the line is coincident with another line.
is_opposite_line	Determine if the line is opposite another line.
parameterization	Get the parametrization of the line.

Properties

origin	Origin of the line.
direction	Direction of the line.

Special methods

__eq__	Equals operator for the `Line` class.

Import detail

```python
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

line	Line being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
tangent	Tangent of the evaluation.
first_derivative	First derivative of the evaluation.
second_derivative	Second derivative of the evaluation.
curvature	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 `trimmed_curve.py` module

Summary

Classes

TrimmedCurve	Represents a trimmed curve.
ReversedTrimmedCurve	Represents a reversed trimmed curve.

TrimmedCurve

class ansys.geometry.core.shapes.curves.trimmed_curve.**TrimmedCurve**(*geometry:* ansys.geometry.core.shapes.curves.curve.Curve, *start:* ansys.geometry.core.math.point.Point3D, *end:* ansys.geometry.core.math.point.Point3D, *interval:* ansys.geometry.core.shapes.parameterization.Inter *length:* *pint.Quantity*, *grpc_client:* ansys.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

evaluate_proportion	Evaluate the curve at a proportional value.
intersect_curve	Get the intersect points of this trimmed curve with another one.

Properties

geometry	Underlying mathematical curve.
start	Start point of the curve.
end	End point of the curve.
length	Calculated length of the edge.
interval	Interval of the curve that provides its boundary.

Special methods

__repr__	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: ansys.geometry.core.shapes.curves.curv start: ansys.geometry.core.math.point.Point3I end: ansys.geometry.core.math.point.Point3I interval: ansys.geometry.core.shapes.parameteriz length: pint.Quantity, grpc_client: ansys.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.

Attributes

temp

Import detail

```
from ansys.geometry.core.shapes.curves.trimmed_curve import ReversedTrimmedCurve
```

Attribute detail

ReversedTrimmedCurve.**temp**

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

cone	Provides for creating and managing a cone.
cylinder	Provides for creating and managing a cylinder.
plane	Provides for creating and managing a cylinder.
sphere	Provides for creating and managing a sphere.
surface	Provides the `Surface` class.
surface_evaluation	Provides for evaluating a surface.
torus	Provides for creating and managing a torus.
trimmed_surface	Provides the `TrimmedSurface` class.

The `cone.py` module

Summary

Classes

Cone	Provides 3D cone representation.
ConeEvaluation	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 |* ansys.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

`contains_param`	Check a parameter is within the parametric range of the surface.
`contains_point`	Check a point is contained by the surface.

Methods

transformed_copy	Create a transformed copy of the cone from a transformation matrix.
mirrored_copy	Create a mirrored copy of the cone along the y-axis.
evaluate	Evaluate the cone at given parameters.
project_point	Project a point onto the cone and evaluate the cone.
parameterization	Parameterize the cone surface as a tuple (U and V respectively).

Properties

origin	Origin of the cone.
radius	Radius of the cone.
half_angle	Half angle of the apex.
dir_x	X-direction of the cone.
dir_y	Y-direction of the cone.
dir_z	Z-direction of the cone.
height	Height of the cone.
surface_area	Surface area of the cone.
volume	Volume of the cone.
apex	Apex point of the cone.
apex_param	Apex parameter of the cone.

Special methods

__eq__	Equals operator for the Cone class.

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 2*pi.

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

cone	Cone being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
normal	Normal to the surface.
u_derivative	First derivative with respect to the U parameter.
v_derivative	First derivative with respect to the V parameter.
uu_derivative	Second derivative with respect to the U parameter.
uv_derivative	Second derivative with respect to the U and V parameters.
vv_derivative	Second derivative with respect to the V parameter.
min_curvature	Minimum curvature of the cone.
min_curvature_direction	Minimum curvature direction.
max_curvature	Maximum curvature of the cone.
max_curvature_direction	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

Cylinder	Provides 3D cylinder representation.
CylinderEvaluation	Provides evaluation of a cylinder at given parameters.

`Cylinder`

class ansys.geometry.core.shapes.surfaces.cylinder.**Cylinder**(*origin: numpy.ndarray | ansys.geometry.core.typing.RealSequence* | ansys.geometry.core.math.point.Point3D, *radius: pint.Quantity* | ansys.geometry.core.misc.measurements.Distance | *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 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

contains_param	Check a parameter is within the parametric range of the surface.
contains_point	Check a point is contained by the surface.

Methods

surface_area	Get the surface area of the cylinder.
volume	Get the volume of the cylinder.
transformed_copy	Create a transformed copy of the cylinder from a transformation matrix.
mirrored_copy	Create a mirrored copy of the cylinder along the y-axis.
evaluate	Evaluate the cylinder at the given parameters.
project_point	Project a point onto the cylinder and evaluate the cylinder.
parameterization	Parameterize the cylinder surface as a tuple (U and V respectively).

Properties

origin	Origin of the cylinder.
radius	Radius of the cylinder.
dir_x	X-direction of the cylinder.
dir_y	Y-direction of the cylinder.
dir_z	Z-direction of the cylinder.

Special methods

__eq__	Equals operator for the `Cylinder` class.

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 2*pi.

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.I

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

cylinder	Cylinder being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
normal	Normal to the surface.
u_derivative	First derivative with respect to the U parameter.
v_derivative	First derivative with respect to the V parameter.
uu_derivative	Second derivative with respect to the U parameter.
uv_derivative	Second derivative with respect to the U and V parameters.
vv_derivative	Second derivative with respect to the V parameter.
min_curvature	Minimum curvature of the cylinder.
min_curvature_direction	Minimum curvature direction.
max_curvature	Maximum curvature of the cylinder.
max_curvature_direction	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
Mminimum 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

PlaneSurface	Provides 3D plane surface representation.
PlaneEvaluation	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

contains_param	Check a ParamUV is within the parametric range of the surface.
contains_point	Check whether a 3D point is in the domain of the plane.

Methods

parameterization	Parametrize the plane.
project_point	Evaluate the plane at a given 3D point.
transformed_copy	Get a transformed version of the plane given the transform.
evaluate	Evaluate the plane at a given u and v parameter.

Properties

origin	Origin of the cylinder.
dir_x	X-direction of the cylinder.
dir_y	Y-direction of the cylinder.
dir_z	Z-direction of the cylinder.

Special methods

__eq__	Check whether two planes are equal.

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 cylinder.

property PlaneSurface.**dir_x**: ***ansys.geometry.core.math.vector.UnitVector3D***

X-direction of the cylinder.

property PlaneSurface.**dir_y**: ***ansys.geometry.core.math.vector.UnitVector3D***

Y-direction of the cylinder.

property PlaneSurface.**dir_z**: ***ansys.geometry.core.math.vector.UnitVector3D***

Z-direction of the cylinder.

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.ParamU

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

plane	Plane being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Point on the surface, based on the evaluation.
normal	Normal to the surface.
u_derivative	First derivative with respect to u.
v_derivative	First derivative with respect to v.
uu_derivative	Second derivative with respect to u.
uv_derivative	Second derivative with respect to u and v.
vv_derivative	Second derivative with respect to v.
min_curvature	Minimum curvature.
min_curvature_direction	Minimum curvature direction.
max_curvature	Maximum curvature.
max_curvature_direction	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 cylinder.

The `sphere.py` module

Summary

Classes

Sphere	Provides 3D sphere representation.
SphereEvaluation	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* | ansys.geometry.core.misc.measurements.Distance | *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 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

contains_param	Check a parameter is within the parametric range of the surface.
contains_point	Check a point is contained by the surface.

Methods

transformed_copy	Create a transformed copy of the sphere from a transformation matrix.
mirrored_copy	Create a mirrored copy of the sphere along the y-axis.
evaluate	Evaluate the sphere at the given parameters.
project_point	Project a point onto the sphere and evaluate the sphere.
parameterization	Parameterization of the sphere surface as a tuple (U, V).

Properties

origin	Origin of the sphere.
radius	Radius of the sphere.
dir_x	X-direction of the sphere.
dir_y	Y-direction of the sphere.
dir_z	Z-direction of the sphere.
surface_area	Surface area of the sphere.
volume	Volume of the sphere.

Special methods

__eq__	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 `2*pi`.

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.Paran

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

sphere	Sphere being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
normal	The normal to the surface.
u_derivative	First derivative with respect to the U parameter.
v_derivative	First derivative with respect to the V parameter.
uu_derivative	Second derivative with respect to the U parameter.
uv_derivative	Second derivative with respect to the U and V parameters.
vv_derivative	Second derivative with respect to the V parameter.
min_curvature	Minimum curvature of the sphere.
min_curvature_direction	Minimum curvature direction.
max_curvature	Maximum curvature of the sphere.
max_curvature_direction	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

Surface	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

parameterization	Parameterize the surface as a tuple (U and V respectively).
contains_param	Check a parameter is within the parametric range of the surface.
contains_point	Check a point is contained by the surface.
transformed_copy	Create a transformed copy of the surface.
__eq__	Determine if two surfaces are equal.
evaluate	Evaluate the surface at the given parameter.
project_point	Project a point to the surface.

Methods

trim	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

SurfaceEvaluation	Provides for evaluating a surface.

SurfaceEvaluation

class ansys.geometry.core.shapes.surfaces.surface_evaluation.**SurfaceEvaluation**(*parameter:* ansys.geometry.core.shapes.param

Provides for evaluating a surface.

Overview

Properties

parameter	Parameter that the evaluation is based upon.
position	Point on the surface, based on the evaluation.
normal	Normal to the surface.
u_derivative	First derivative with respect to the U parameter.
v_derivative	First derivative with respect to the V parameter.
uu_derivative	Second derivative with respect to the U parameter.
uv_derivative	The second derivative with respect to the U and V parameters.
vv_derivative	The second derivative with respect to v.
min_curvature	Minimum curvature.
min_curvature_direction	Minimum curvature direction.
max_curvature	Maximum curvature.
max_curvature_direction	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

Torus	Provides 3D torus representation.
TorusEvaluation	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* | 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*, *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 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

contains_param	Check a parameter is within the parametric range of the surface.
contains_point	Check a point is contained by the surface.

Methods

transformed_copy	Create a transformed copy of the torus from a transformation matrix.
mirrored_copy	Create a mirrored copy of the torus along the y-axis.
evaluate	Evaluate the torus at the given parameters.
parameterization	Parameterize the torus surface as a tuple (U and V respectively).
project_point	Project a point onto the torus and evaluate the torus.

Properties

origin	Origin of the torus.
major_radius	Semi-major radius of the torus.
minor_radius	Semi-minor radius of the torus.
dir_x	X-direction of the torus.
dir_y	Y-direction of the torus.
dir_z	Z-direction of the torus.
volume	Volume of the torus.
surface_area	Surface_area of the torus.

Special methods

`__eq__`	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 `2*pi`.

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.ParamU

Bases: *ansys.geometry.core.shapes.surfaces.surface_evaluation.SurfaceEvaluation*

Evaluate the torus`` at given parameters.

Parameters

Torus: ~ansys.geometry.core.shapes.surfaces.torus.Torus
Torust to evaluate.

parameter: ParamUV
Parameters (u, v) to evaluate the torus at.

Overview

Properties

torus	Torus being evaluated.
parameter	Parameter that the evaluation is based upon.
position	Position of the evaluation.
normal	Normal to the surface.
u_derivative	First derivative with respect to the U parameter.
v_derivative	First derivative with respect to the V parameter.
uu_derivative	Second derivative with respect to the U parameter.
uv_derivative	Second derivative with respect to the U and V parameters.
vv_derivative	Second derivative with respect to the V parameter.
curvature	Curvature of the torus.
min_curvature	Minimum curvature of the torus.
min_curvature_direction	Minimum curvature direction.
max_curvature	Maximum curvature of the torus.
max_curvature_direction	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 detail

TorusEvaluation.**cache**

Description

Provides for creating and managing a torus.

The `trimmed_surface.py` module

Summary

Classes

TrimmedSurface	Represents a trimmed surface.
ReversedTrimmedSurface	Represents a reversed trimmed surface.

`TrimmedSurface`

class ansys.geometry.core.shapes.surfaces.trimmed_surface.**TrimmedSurface**(*geometry:* ansys.geometry.core.shapes.surfaces.surfa *box_uv:* ansys.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

get_proportional_parameters	Convert non-proportional parameters into proportional parameters.
normal	Provide the normal to the surface.
project_point	Project a point onto the surface and evaluate it at that location.
evaluate_proportion	Evaluate the surface at proportional u and v parameters.

Properties

geometry	Underlying mathematical surface.
box_uv	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.sur *box_uv:* an-sys.geometry.core.shapes.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

normal	Provide the normal to the surface.
project_point	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

BoxUV	Provides the implementation for `BoxUV` class.

Enums

LocationUV	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

is_empty	Check if this BoxUV is empty.
proportion	Evaluate the BoxUV at the given proportions.
get_center	Evaluate the this BoxUV in the center.
is_negative	Check whether the BoxUV is negative.
contains	Check whether the BoxUV contains a given u and v pair parameter.
inflate	Enlarge the BoxUV u and v intervals by deltas.
get_corner	Get the corner location of the BoxUV.

Properties

interval_u	u interval.
interval_v	v interval.

Special methods

__eq__	Check whether two BoxUV instances are equal.
__ne__	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*, ***kwds*)

Bases: enum.Enum

Provides the enumeration for indicating locations for BoxUV.

Overview

Attributes

TopLeft
TopCenter
TopRight
BottomLeft
BottomCenter
BottomRight
LeftCenter
RightCenter
Center

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

ParamUV	Parameter class containing 2 parameters: (u, v).
Interval	Interval class that defines a range of values.
Parameterization	Parameterization class describes the parameters of a specific geometry.

Enums

ParamForm	ParamForm enum class that defines the form of a Parameterization.
ParamType	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

__add__	Add the u and v components of the other ParamUV to this ParamUV.
__sub__	Subtract the u and v components of a ParamUV from this ParamUV.
__mul__	Multiplies the u and v components of this ParamUV by a ParamUV.
__truediv__	Divides the u and v components of this ParamUV by a ParamUV.
__iter__	Iterate a `ParamUV`.
__repr__	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

is_open	If the interval is open (-inf, inf).
is_closed	If the interval is closed. Neither value is inf or -inf.
is_empty	Check if the current interval is empty.
get_span	Get the quantity contained by the interval.
get_relative_val	Get an evaluation property of the interval, used in BoxUV.
is_negative	Indicate whether the current interval is negative.
self_unite	Get the union of two intervals and update the current interval.
self_intersect	Get the intersection of two intervals and update the current one.
contains_value	Check if the current interval contains the value `t`.
contains	Check if interval contains value `t` using default accuracy.
inflate	Enlarge the current interval by the given delta value.

Properties

start	Start value of the interval.
end	End value of the interval.

Attributes

not_empty

Static methods

unite	Get the union of two intervals.
intersect	Get the intersection of two intervals.

Special methods

__eq__	Compare two intervals.
__repr__	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

form	The form of the parameterization.
type	The type of the parameterization.
interval	The interval of the parameterization.

Special methods

__repr__	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*, ***kwds*)

Bases: enum.Enum

ParamForm enum class that defines the form of a Parameterization.

Overview

Attributes

OPEN
CLOSED
PERIODIC
OTHER

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*, ***kwds*)

Bases: enum.Enum

ParamType enum class that defines the type of a Parameterization.

Overview

Attributes

LINEAR
CIRCULAR
OTHER

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

arc	Provides for creating and managing an arc.
box	Provides for creating and managing a box (quadrilateral).
circle	Provides for creating and managing a circle.
edge	Provides for creating and managing an edge.
ellipse	Provides for creating and managing an ellipse.
face	Provides for creating and managing a face (closed 2D sketch).
gears	Module for creating and managing gears.
polygon	Provides for creating and managing a polygon.
segment	Provides for creating and managing a segment.
sketch	Provides for creating and managing a sketch.
slot	Provides for creating and managing a slot.
trapezoid	Provides for creating and managing a trapezoid.
triangle	Provides for creating and managing a triangle.

The arc.py module

Summary

Classes

Arc	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 spands the clockwise angle.

Overview

Constructors

from_three_points	Create an arc from three given points.
from_start_end_and_radius	Create an arc from a starting point, an ending point, and a radius.
from_start_center_and_angle	Create an arc from a starting point, a center point, and an angle.

Properties

start	Starting point of the arc line.
end	Ending point of the arc line.
center	Center point of the arc.
length	Length of the arc.
radius	Radius of the arc.
angle	Angle of the arc.
is_clockwise	Flag indicating whether the rotation of the angle is clockwise.
sector_area	Area of the sector of the arc.
visualization_polydata	VTK polydata representation for PyVista visualization.

Attributes

to_start_vector
to_end_vector

Special methods

__eq__	Equals operator for the Arc class.
__ne__	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.

Attribute detail

Arc.**to_start_vector**

Arc.**to_end_vector**

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 spands 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

center	Center point of the box.
width	Width of the box.
height	Height of the box.
perimeter	Perimeter of the box.
area	Area of the box.
visualization_polydata	VTK polydata representation for PyVista visualization.

Attributes

angle
rotation
width_magnitude
height_magnitude
half_h
half_w
corner_1
corner_2
corner_3
corner_4

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.

Attribute detail

Box.**angle**

Box.**rotation**

Box.**width_magnitude**

Box.**height_magnitude**

Box.**half_h**

Box.**half_w**

Box.**corner_1**

Box.**corner_2**

Box.**corner_3**

Box.**corner_4**

Description

Provides for creating and managing a box (quadrilateral).

The `circle.py` module

Summary

Classes

SketchCircle	Provides for modeling a circle.

SketchCircle

class ansys.geometry.core.sketch.circle.**SketchCircle**(*center:* ansys.geometry.core.math.point.Point2D, *radius:* *pint.Quantity* | ansys.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

plane_change	Redefine the plane containing the SketchCircle objects.

Properties

center	Center of the circle.
perimeter	Perimeter of the circle.
visualization_polydata	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

SketchEdge	Provides for modeling edges forming sketched shapes.

SketchEdge

class ansys.geometry.core.sketch.edge.**SketchEdge**

Provides for modeling edges forming sketched shapes.

Overview

Methods

plane_change	Redefine the plane containing SketchEdge objects.

Properties

start	Starting point of the edge.
end	Ending point of the edge.
length	Length of the edge.
visualization_polydata	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 metho 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

SketchEllipse	Provides for modeling an ellipse.

`SketchEllipse`

class ansys.geometry.core.sketch.ellipse.**SketchEllipse**(*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*, *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

plane_change	Redefine the plane containing SketchEllipse objects.

Properties

center	Center point of the ellipse.
angle	Orientation angle of the ellipse.
perimeter	Perimeter of the circle.
visualization_polydata	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 SkethFace.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

SketchFace	Provides for modeling a face.

SketchFace

`class ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling a face.

Overview

Methods

plane_change	Redefine the plane containing `SketchFace` objects.

Properties

edges	List of all component edges forming the face.
perimeter	Perimeter of the face.
visualization_polydata	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

Gear	Provides the base class for sketching gears.
DummyGear	Provides the dummy class for sketching gears.
SpurGear	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

visualization_polydata	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.

Overview

Attributes

outer_radius
inner_radius
repeat_angle
tooth_angle
gap_angle

Import detail

```python
from ansys.geometry.core.sketch.gears import DummyGear
```

Attribute detail

DummyGear.**outer_radius**

DummyGear.**inner_radius**

DummyGear.**repeat_angle**

DummyGear.**tooth_angle**

DummyGear.**gap_angle**

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

pressure_angle	Pressure angle of the spur gear.
origin	Origin of the spur gear.
module	Module of the spur gear.
n_teeth	Number of teeth of the spur gear.
ref_diameter	Reference diameter of the spur gear.
base_diameter	Base diameter of the spur gear.
addendum	Addendum of the spur gear.
dedendum	Dedendum of the spur gear.
tip_diameter	Tip diameter of the spur gear.
root_diameter	Root diameter of the spur gear.

Import detail

```
from ansys.geometry.core.sketch.gears import SpurGear
```

Property detail

property SpurGear.**pressure_angle: pint.Quantity**

Pressure angle of the spur gear.

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.**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

Polygon	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

center	Center point of the polygon.
inner_radius	Inner radius (apothem) of the polygon.
n_sides	Number of sides of the polygon.
angle	Orientation angle of the polygon.
length	Side length of the polygon.
outer_radius	Outer radius of the polygon.
perimeter	Perimeter of the polygon.
area	Area of the polygon.
visualization_polydata	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

SketchSegment	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

plane_change	Redefine the plane containing SketchSegment objects.

Properties

start	Starting point of the segment.
end	Ending point of the segment.
length	Length of the segment.
visualization_polydata	VTK polydata representation for PyVista visualization.

Special methods

__eq__	Equals operator for the SketchSegment class.
__ne__	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

Sketch	Provides for building 2D sketch elements.

Attributes

SketchObject	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

translate_sketch_plane	Translate the origin location of the active sketch plane.
translate_sketch_plane_by_offset	Translate the origin location of the active sketch plane by offsets.
translate_sketch_plane_by_distance	Translate the origin location active sketch plane by distance.
get	Get a list of shapes with a given tag.
face	Add a sketch face to the sketch.
edge	Add a sketch edge to the sketch.
select	Add all objects that match provided tags to the current context.
segment	Add a segment sketch object to the sketch plane.
segment_to_point	Add a segment to the sketch plane starting from the previous end point.
segment_from_point_and_vector	Add a segment to the sketch starting from a given starting point.
segment_from_vector	Add a segment to the sketch starting from the previous end point.
arc	Add an arc to the sketch plane.
arc_to_point	Add an arc to the sketch starting from the previous end point.
arc_from_three_points	Add an arc to the sketch plane from three given points.
arc_from_start_end_and_radius	Add an arc from the start, end points and a radius.
arc_from_start_center_and_angle	Add an arc from the start, center point, and angle.
triangle	Add a triangle to the sketch using given vertex points.
trapezoid	Add a trapezoid to the sketch using given vertex points.

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circle	Add a circle to the plane at a given center.
box	Create a box on the sketch.
slot	Create a slot on the sketch.
ellipse	Create an ellipse on the sketch.
polygon	Create a polygon on the sketch.
dummy_gear	Create a dummy gear on the sketch.
spur_gear	Create a spur gear on the sketch.
tag	Add a tag to the active selection of sketch objects.
plot	Plot all objects of the sketch to the scene.
plot_selection	Plot the current selection to the scene.
sketch_polydata	Get polydata configuration for all objects of the sketch.
sketch_polydata_faces	Get polydata configuration for all faces of the sketch to the scene.
sketch_polydata_edges	Get polydata configuration for all edges of the sketch to the scene.

Properties

plane	Sketch plane configuration.
edges	List of all independently sketched edges.
faces	List of all independently sketched faces.

Import detail

```python
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_trame: 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_trame
[bool, default: `None`] Whether to enables the use of trame. The default is `None`, in which case the `ansys.tools.visualization_interface.USE_TRAME` 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_trame: 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_trame
[bool, default: `None`] Whether to enables the use of trame. The default is `None`, in which case the `ansys.tools.visualization_interface.USE_TRAME` 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

Slot	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

center	Center of the slot.
width	Width of the slot.
height	Height of the slot.
perimeter	Perimeter of the slot.
area	Area of the slot.
visualization_polydata	VTK polydata representation for PyVista visualization.

Attributes

width_magnitude
height_magnitude
angle
rotation
half_h
half_box_w
corner_1
corner_2
corner_3
corner_4
arc_1_center
arc_2_center

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.

Attribute detail

Slot.**width_magnitude**

Slot.**height_magnitude**

Slot.**angle**

Slot.**rotation**

Slot.**half_h**

Slot.**half_box_w**

Slot.**corner_1**

Slot.**corner_2**

Slot.**corner_3**

Slot.**corner_4**

Slot.**arc_1_center**

Slot.**arc_2_center**

Description

Provides for creating and managing a slot.

The `trapezoid.py` module

Summary

Classes

Trapezoid	Provides for modeling a 2D trapezoid.

Trapezoid

class ansys.geometry.core.sketch.trapezoid.**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*)

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

center	Center of the trapezoid.
base_width	Width of the trapezoid.
height	Height of the trapezoid.
visualization_polydata	VTK polydata representation for PyVista visualization.

Attributes

width_magnitude
height_magnitude
angle
base_angle
base_offset_right
base_offset_left
rotation
half_h
half_w
rotated_point_1
rotated_point_2
rotated_point_3
rotated_point_4

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.

Attribute detail

Trapezoid.**width_magnitude**

Trapezoid.**height_magnitude**

Trapezoid.**angle**

Trapezoid.**base_angle**

Trapezoid.**base_offset_right**

Trapezoid.**base_offset_left**

Trapezoid.**rotation**

Trapezoid.**half_h**

Trapezoid.**half_w**

Trapezoid.**rotated_point_1**

Trapezoid.**rotated_point_2**

Trapezoid.**rotated_point_3**

Trapezoid.**rotated_point_4**

Description

Provides for creating and managing a trapezoid.

The `triangle.py` module

Summary

Classes

Triangle	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

point1	Triangle vertex 1.
point2	Triangle vertex 2.
point3	Triangle vertex 3.
visualization_polydata	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

measurement_tools	Provides tools for measurement.
prepare_tools	Provides tools for preparing geometry for use with simulation.
problem_areas	The problem area definition.
repair_tool_message	Module for repair tool message.
repair_tools	Provides tools for repairing bodies.

The `measurement_tools.py` module

Summary

Classes

Gap	Represents a gap between two bodies.
MeasurementTools	Measurement tools for PyAnsys Geometry.

Gap

class ansys.geometry.core.tools.measurement_tools.**Gap**(*distance:* ansys.geometry.core.misc.measurements.Distance)

Represents a gap between two bodies.

Parameters

distance
[Distance] Distance between two sides of the gap.

Overview

Properties

distance	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: ansys.geometry.core.connection.GrpcClient*)

Measurement tools for PyAnsys Geometry.

Parameters

grpc_client
[GrpcClient] gRPC client to use for the measurement tools.

Overview

Methods

min_distance_between_objects	Find the gap between two bodies.

Import detail

```
from ansys.geometry.core.tools.measurement_tools import MeasurementTools
```

Method detail

`MeasurementTools.`**`min_distance_between_objects`**(*body1:* ansys.geometry.core.designer.body.Body, *body2:* ansys.geometry.core.designer.body.Body) → *Gap*

Find the gap between two bodies.

Parameters

body1
[Body] First body to measure the gap.

body2
[Body] Second body to measure the gap.

Returns

Gap
Gap between two bodies.

Description

Provides tools for measurement.

The `prepare_tools.py` module

Summary

Classes

PrepareTools	Prepare tools for PyAnsys Geometry.

`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

extract_volume_from_faces	Extract a volume from input faces.
extract_volume_from_edge_loops	Extract a volume from input edge loops.
share_topology	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*]*) → 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]] 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.

Description

Provides tools for preparing geometry for use with simulation.

The problem_areas.py module

Summary

Classes

ProblemArea	Represents problem areas.
DuplicateFaceProblemAreas	Provides duplicate face problem area definition.
MissingFaceProblemAreas	Provides missing face problem area definition.
InexactEdgeProblemAreas	Represents an inexact edge problem area with unique identifier and associated edges.
ExtraEdgeProblemAreas	Represents a extra edge problem area with unique identifier and associated edges.
ShortEdgeProblemAreas	Represents a short edge problem area with a unique identifier and associated edges.
SmallFaceProblemAreas	Represents a small face problem area with a unique identifier and associated faces.
SplitEdgeProblemAreas	Represents a split edge problem area with unique identifier and associated edges.
StitchFaceProblemAreas	Represents a stitch face problem area with unique identifier and associated faces.

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

```python
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: ansys.geometry.core.connection.GrpcClient, faces: list[*ansys.geometry.core.designer.face.Fa

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 body.

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 the edges 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 the edges 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 body.

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 the 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 the 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 body.

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 the 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 the 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 body.

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 the ids of the 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 the ids of the 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 body.

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 the ids of the 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 the ids of the 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 body.

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

faces
[list[Face]] List of edges associated with the design.

Overview

Methods

fix	Fix the problem area.

Properties

faces	The list of the ids of the edges 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 the ids of the edges 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 body.

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 body.

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

bodies
[list[Body]] List of bodies associated with the design.

Overview

Methods

fix	Fix the problem area.

Properties

bodies	The list of the 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 the 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.

Description

The problem area definition.

The `repair_tool_message.py` module

Summary

Classes

RepairToolMessage	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]*)

Provides return message for the repair tool methods.

Overview

Properties

success	The success of the repair operation.
created_bodies	The list of the created bodies after the repair operation.
modified_bodies	The list of the modified bodies after 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.

Description

Module for repair tool message.

The repair_tools.py module

Summary

Classes

RepairTools	Repair tools for PyAnsys Geometry.

RepairTools

class ansys.geometry.core.tools.repair_tools.**RepairTools**(*grpc_client: ansys.geometry.core.connection.GrpcClient*)

Repair tools for PyAnsys Geometry.

Overview

Methods

find_split_edges	Find split edges in the given list of bodies.
find_extra_edges	Find the extra edges in the given list of bodies.
find_inexact_edges	Find inexact edges in the given list of bodies.
find_short_edges	Find the short edge problem areas.
find_duplicate_faces	Find the duplicate face problem areas.
find_missing_faces	Find the missing faces.
find_small_faces	Find the small face problem areas.
find_stitch_faces	Return the list of stitch face problem areas.

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.

Description

Provides tools for repairing bodies.

Description

PyAnsys Geometry tools subpackage.

The `errors.py` module

Summary

Exceptions

GeometryRuntimeError	Provides error message when Geometry service passes a runtime error.
GeometryExitedError	Provides error message to raise when Geometry service has exited.

Functions

handler	Pass signal to the custom interrupt handler.
protect_grpc	Capture gRPC exceptions and raise a more succinct error message.

Constants

SIGINT_TRACKER

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

PyGeometryCustomAdapter	Keeps the reference to the Geometry service instance name dynamic.
PyGeometryPercentStyle	Provides a common messaging style.
PyGeometryFormatter	Provides a `Formatter` class for overwriting default format styles.
InstanceFilter	Ensures that the `instance_name` record always exists.
Logger	Provides the logger used for each PyAnsys Geometry session.

Functions

addfile_handler	Add a file handler to the input.
add_stdout_handler	Add a standout handler to the logger.

Attributes

string_to_loglevel

Constants

LOG_LEVEL
FILE_NAME
DEBUG
INFO
WARN
ERROR
CRITICAL
STDOUT_MSG_FORMAT
FILE_MSG_FORMAT
DEFAULT_STDOUT_HEADER
DEFAULT_FILE_HEADER
NEW_SESSION_HEADER
LOG

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

process	Process the logging message and keyword arguments passed in to
log_to_file	Add a file handler to the logger.
log_to_stdout	Add a standard output handler to the logger.
setLevel	Change the log level of the object and the attached handlers.

Attributes

level
file_handler
stdout_handler
logger
std_out_handler

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

filter	Ensure that the instance_name 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

log_to_file	Add a file handler to the logger.
log_to_stdout	Add the standard output handler to the logger.
setLevel	Change the log level of the object and the attached handlers.
add_child_logger	Add a child logger to the main logger.
add_instance_logger	Add a logger for a Geometry service instance.
add_handling_uncaught_expections	Redirect the output of an exception to a logger.

Attributes

file_handler
std_out_handler
logger
propagate
level
debug
info
warning
error
critical
log

Special methods

__getitem__	Overload the access method by item for the Logger class.

Import detail

```
from ansys.geometry.core.logger import Logger
```

Attribute detail

Logger.**file_handler = None**

Logger.**std_out_handler = None**

Logger.**logger**

Logger.**propagate = True**

Logger.**level**

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_expections`(*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.")

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 standout 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`**

`logger.`**`FILE_NAME = 'pyansys-geometry.log'`**

`logger.`**`DEBUG`**

`logger.`**`INFO`**

`logger.`**`WARN`**

logger.**ERROR**

logger.**CRITICAL**

logger.**STDOUT_MSG_FORMAT = '%(levelname)s - %(instance_name)s - %(module)s - %(funcName)s - %(message)s'**

logger.**FILE_MSG_FORMAT**

logger.**DEFAULT_STDOUT_HEADER = Multiline-String**

```
"""
LEVEL - INSTANCE NAME - MODULE - FUNCTION - MESSAGE
"""
```

logger.**DEFAULT_FILE_HEADER**

logger.**NEW_SESSION_HEADER**

logger.**LOG**

logger.**string_to_loglevel**

The `modeler.py` module

Summary

Classes

Modeler	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:* ansys.geometry.core.connection.docker_instance.LocalDockerInstance | *None = None*, *product_instance:* ansys.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
[Union[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

create_design	Initialize a new design with the connected client.
get_active_design	Get the active design on the modeler object.
read_existing_design	Read the existing design on the service with the connected client.
close	Access the client's close method.
exit	Access the client's close method.
open_file	Open a file.
run_discovery_script_file	Run a Discovery script file.

Properties

client	Modeler instance client.
designs	All designs within the modeler workspace.
repair_tools	Access to repair tools.
prepare_tools	Access to prepare tools.
measurement_tools	Access to measurement tools.

Special methods

__repr__	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.**designs**: dict[str, ***ansys.geometry.core.designer.design.Design***]

All designs within the modeler workspace.

Notes

This property is read-only. **DO NOT** modify the dictionary.

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.

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_designs: bool = True*`)` → None

Access the client's close method.

Parameters

close_designs
[bool, default: `True`] Whether to close all designs before closing the client.

`Modeler.`**`exit`**`(`*close_designs: bool = True*`)` → None

Access the client's close method.

Parameters

close_designs
[bool, default: `True`] Whether to close all designs 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, `.scdocx` and HOOPS Exchange formats are supported. On Linux, only the `.scdocx` 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 23.2.1 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 23.2 can be passed as an integer 232, a string "232" 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.

Description

Provides for interacting with the Geometry service.

The `typing.py` module

Summary

Attributes

Real	Type used to refer to both integers and floats as possible values.
RealSequence	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 = 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 you will ONLY be working with one design.

`core.DOCUMENTATION_BUILD: bool`

Global flag for the documentation to use the proper PyVista Jupyter backend.

`core.__version__`

PyAnsys Geometry version.

CHAPTER

EIGHT

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

Download this example as a Jupyter Notebook or as a Python script.

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])

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())
```

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```
print("\nNew magnitude:")
print(vec_3.normalize().magnitude)
```

```
Magnitude of vec_3:
601.4694173438911

Normalized vec_3:
[-0.03009297 -0.99705818 -0.07049402]

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]:
```python
plane = Plane(Point3D([0,0,0])) # XY plane

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]:
```python
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]:
```python
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]:
```python
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}.")

# 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]:
```python
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}")
```

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```
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.].
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

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 7.000000000000001 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))
```

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```
      .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-...
```

Download this example

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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
```

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```
modeler = launch_modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x1c625093f20)

Ansys Geometry Modeler Client (0x1c61141e240)
  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)
```

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```
    .circle(center=Point2D([3, -4], unit=UNITS.m), radius=outer_hole_radius)
)

# Plot the sketch
sketch.plot()
```

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

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&quot;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 54.28672587712815 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
```

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```
# 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,
    [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=&quot;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=&quot;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 serssion 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 import Modeler
from ansys.geometry.core.connection.defaults import GEOMETRY_SERVICE_DOCKER_IMAGE
from ansys.geometry.core.connection.docker_instance import LocalDockerInstance
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 (0x24de661f3e0)

Ansys Geometry Modeler Client (0x24de669fb00)
  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)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪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)
```

Render in different colors

You can render the objects in different colors automatically using PyVista's default color cycler. In order to do this, activate the `multi_colors=True` option when calling the `plot()` method.

In the following cell we will create a new design and plot a prism and a cylinder in different colors.

```
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)
```

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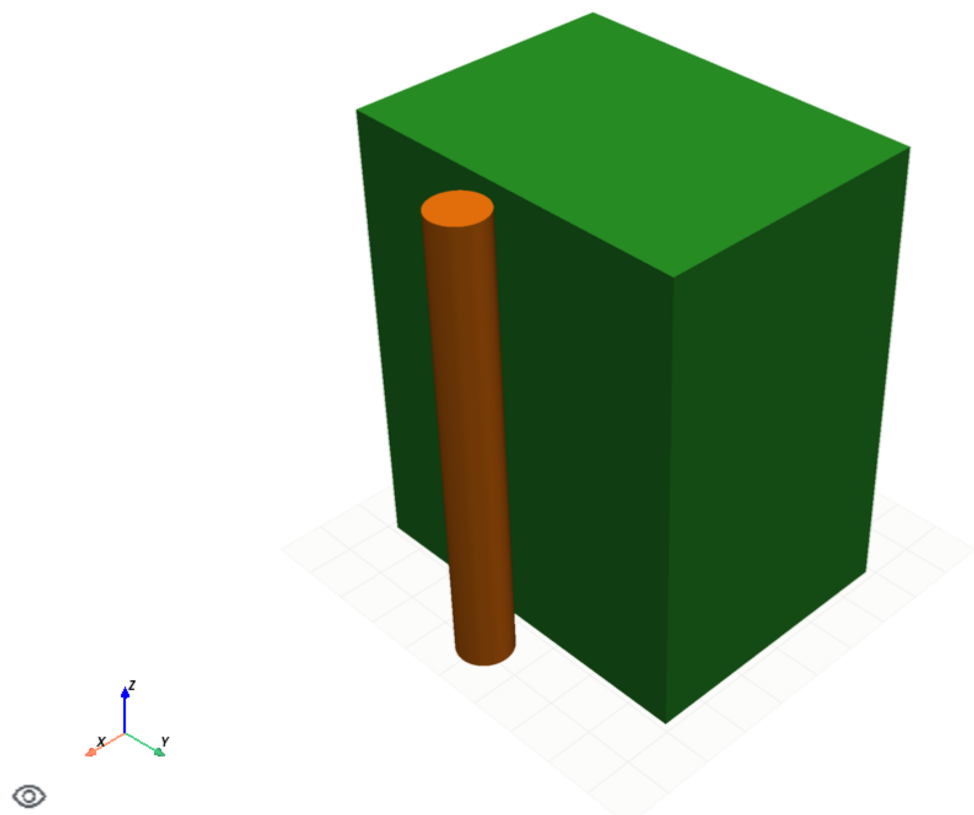
```
# Design plotting
design.plot(multi_colors=True)
```



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)
```

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```
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)
```

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

```
None
```

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.

[13]:
```
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

Download this example as a Jupyter Notebook or as a Python script.

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(
    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()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪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()
```

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

Instantiate the modeler

Launch a modeling service and connect to it.

[3]:
```python
modeler = launch_modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x1758100aab0)

Ansys Geometry Modeler Client (0x17581cff470)
  Target:     localhost:700
  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]:
```python
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]:
```python
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 (0x1cdd4bc57f0)

Ansys Geometry Modeler Client (0x1cdd44eecf0)
  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)
```

(continues on next page)

(continued from previous page)

```
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=&quot;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=&quot;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=&quot;Content-...
```

After creating the sketch, extrude it.

```
[5]: # Create a design
design = modeler.create_design("AdvancedFeatures_SpurGear")

# 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,␣
↪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 0x1fb1ae1c5f0>
```

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 (0x1fb2eac68a0)

Ansys Geometry Modeler Client (0x1fb2e95cc80)
  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,
```

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```
    [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))

# 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=&quot;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 :ref:`Sketch <ref_sketch>` subpackage.

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 profle 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 0x222ba1a7a40>
```

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]:
```python
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]:
```python
# 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)
    )

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=&quot;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]:
```python
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=&quot;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=&quot;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: 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]:
```python
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]:
```python
dataset = comp.tessellate(merge_bodies=True)
dataset
```

```
C:\Users\ansys\actions-runner\_work\pyansys-geometry\pyansys-geometry\.venv\Lib\site-
↪packages\pyvista\core\pointset.py:843: PyVistaDeprecationWarning: `PolyData`␣
↪constructor parameter `n_faces` is deprecated and no longer used.
  warnings.warn(
```

[3]:
```
MultiBlock (0x1b8d54ef1c0)
  N Blocks    1
  X Bounds    -25.000, 25.000
  Y Bounds    -24.999, 34.999
  Z Bounds    0.000, 20.000
```

If you want to tessellate the body and return the geometry as triangles, single body tessellation is possible. If you want to merge the individual faces of the tessellation, enable the `merge` option so that the body is rendered into a single mesh. This preserves the number of triangles and only merges the topology.

Code without merging the body

[4]:
```
dataset = body.tessellate()
dataset
```

[4]:
```
MultiBlock (0x1b8d54ef640)
  N Blocks    7
  X Bounds    -25.000, 25.000
  Y Bounds    -14.999, 34.999
  Z Bounds    10.000, 20.000
```

Code with merging the body

[5]:
```
mesh = body.tessellate(merge=True)
mesh
```

[5]:
```
PolyData (0x1b8d574da80)
  N Cells:    1640
  N Points:   1650
  N Strips:   0
  X Bounds:   -2.500e+01, 2.500e+01
  Y Bounds:   -1.500e+01, 3.500e+01
  Z Bounds:   1.000e+01, 2.000e+01
  N Arrays:   0
```

Plot design

Plot the design.

[6]:
```
design.plot()
```

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

Close the modeler

Close the modeler.

[7]:
```
modeler.close()
```

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8.3.4 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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8.3.5 Modeling: Boolean operations

This example shows how to use Boolean operations for geometry manipulation.

Perform required imports

Perform the required imports.

[1]:
```
from typing import List

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 (0x1cbd4f2e060)

Ansys Geometry Modeler Client (0x1cbd4f2df70)
  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)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&quot;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 0x1cbd557b350
  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=&quot;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 0x1cbd7725df0
  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)
```

```
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↪http-equiv=&quot;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 0x1cbd6a76210
  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")

# 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)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&quot;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 0x1cbd69e8e60
```

(continues on next page)

(continued from previous page)

```
  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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8.3.6 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,
```

(continues on next page)

(continued from previous page)

```
  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 (0x1914c8acaa0)

Ansys Geometry Modeler Client (0x1915f1a1850)
  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 a 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 will be 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()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪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-...
```

Halfing 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()
(
   asymmetric_profile.segment(Point2D([1, 1]), Point2D([-1, 1]))
   .segment_to_point(Point2D([0, 0.5]))
   .segment_to_point(Point2D([-1, -1]))
   .segment_to_point(Point2D([1, -1]))
   .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=&quot;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=&quot;Content-...
```

Mirror the triangular body

We will first make a copy of the triangular body, then using `mirror()`, we will 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=&quot;Content-...
```

Closing the modeler

[18]:
```
m.close()
```

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8.3.7 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,
  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 (0x1f4e07dc080)

Ansys Geometry Modeler Client (0x1f4f44d52b0)
  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)

profile.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪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("bowlsweep", 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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>
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8.3.8 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 (0x1f21e2606e0)

Ansys Geometry Modeler Client (0x1f230ffca40)
  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)
```

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```
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,
    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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8.3.9 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()

# 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)
```

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```
        # 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_scdocx()
print(f"Design exported to {file_location}")
```

```
Design exported to C:\Users\ansys\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↪doc\source\examples\03_modeling\ExportDesignExample.scdocx
```

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_scdocx(download_dir)
print(f"Design exported to {file_location}")
```

```
Design exported to C:\Users\ansys\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↪doc\source\examples\03_modeling\downloads\ExportDesignExample.scdocx
```

Export to SCDOCX format

[4]:
```
# Export the design to a file in the requested directory
file_location = design.export_to_scdocx(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\ansys\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↪doc\source\examples\03_modeling\downloads\ExportDesignExample.scdocx
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\ansys\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\ansys\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\ansys\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\ansys\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\ansys\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\ansys\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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8.3.10 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, we 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)
```

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```
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"
# will have a body associated.
#

# 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
:   |---(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 will be 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
|---(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
--------
```

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```
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
-------
(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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8.3.11 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]:
```python
import ansys.geometry.core as pyansys_geometry

from ansys.geometry.core import 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]:
```python
sketch = Sketch()
sketch.box(Point2D([0, 0]), 10, 10)
```

[2]:
```
<ansys.geometry.core.sketch.sketch.Sketch at 0x1b27989d460>
```

Initiate design on server

Establish a server connection and initiate a design on the server.

[3]:
```python
modeler = Modeler()
design = modeler.create_design("ServiceColors")
```

Extrude the box sketch to create the matrix style design

Given our initial sketch, we will extrude it to create a matrix style design. We will create a 2x3 matrix of bodies. Each body will be separated by 30 units in the X direction and 30 units in the Y direction. We will have a total of 6 bodies.

[4]:
```python
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 our previous design, we will assign a color to each body. We will assign a different color to each one of them. We could have done this assignment while creating the bodies, but we will do it now for the sake of encapsulating the color assignment logic.

[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 will **not** use the colors assigned to the bodies. To plot the design with the assigned colors, we 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 we 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 we pass the parameter `use_service_colors=True` to the plot method, the plot will be 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 we set the global parameter to `True`, the plot will be 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
```

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```
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, we will just demonstrate 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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8.4 Applied examples

These examples demonstrate the usage of PyAnsys Geometry for real-world applications.

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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]:
```python
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.
    """
    # Check if the number is a string
    if isinstance(number, str):
        number = int(number)

    # Calculate the NACA parameters
    m = number // 1000 * 0.01
```

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```
    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
        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=&quot;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()

# Create the design
design = modeler.create_design(f"NACA_Airfoil_{NACA_AIRFOIL}")

```

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```
# 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\ansys\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])
```

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```
# 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=&quot;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=&quot;Content-...
```

Save the design

In this case, the design is saved as an SCDOCX file.

[11]:
```
# Save the design
file = design.export_to_scdocx()
print(f"Design saved to {file}")
```

```
Design saved to C:\Users\ansys\actions-runner\_work\pyansys-geometry\pyansys-geometry\
↪doc\source\examples\04_applied\NACA_Airfoil_6412.scdocx
```

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]:
```python
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=&quot;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]:
```python
from ansys.geometry.core.math import Point2D, Plane, Point3D
from ansys.geometry.core.sketch import Sketch

BOX_LENGTH = 10  # X-Axis
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]:
```python
# 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 0x1b47b6a7650>
```

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]:
```python
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]:
```python
# Save the design
file = design.export_to_pmdb()
print(f"Design saved to {file}.")
```

```
Design saved to C:\Users\ansys\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]:
```python
modeler.close()
```

Download this example

Download this example as a Jupyter Notebook or as a Python script.

CHAPTER

NINE

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 Run tests

PyAnsys Geometry uses pytest for testing.

9.6.1 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"
```

9.6.2 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
```

CHAPTER

TEN

ASSETS

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 3.12. 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-v0.8.dev0-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

10.2.2 Windows

- Windows wheelhouse for Python 3.10
- Windows wheelhouse for Python 3.11
- Windows wheelhouse for Python 3.12

10.2.3 MacOS

- MacOS wheelhouse for Python 3.10
- MacOS wheelhouse for Python 3.11
- MacOS wheelhouse for Python 3.12

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

CHAPTER

ELEVEN

RELEASE NOTES

This document contains the release notes for the PyAnsys Geometry project.

11.1 0.7.2 - 2024-09-11

11.1.1 Added

- allow for platform input when using Ansys Lab #1416
- ensure GrpcClient class closure upon deletion #1417

11.1.2 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

11.1.3 Documentation

- remove title from landing page #1408
- adapt examples to use launch_modeler instead of Modeler obj connection #1410

11.1.4 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

11.1.5 Maintenance

- update CHANGELOG for v0.7.1 #1407
- pre-commit automatic update #1418

11.2 0.7.1 - 2024-09-06

11.2.1 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

11.2.2 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

11.2.3 Documentation

- add project logo #1405

11.2.4 Fixed

- remove `server_logs_folder` argument for Discovery and SpaceClaim #1387

11.2.5 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

11.3 0.7.0 - 2024-08-13

11.3.1 Added

- build: drop support for Python 3.9 #1341
- feat: adapting beartype typehints to +Python 3.10 standard #1347

11.3.2 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

11.3.3 Documentation

- feat: update conf for version 1.x of ansys-sphinx-theme #1351

11.3.4 Fixed

- trapezoid signature change and internal checks #1354

11.3.5 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

11.3.6 Miscellaneous

- chore: pre-commit automatic update #1345

11.4 0.6.6 - 2024-08-01

11.4.1 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

11.4.2 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

11.4.3 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

11.4.4 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 jupytext 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

11.4.5 Miscellaneous

- chore: pre-commit automatic update #1327, #1333

11.5 0.6.5 - 2024-07-02

11.5.1 Changed

- chore: update CHANGELOG for v0.6.4 #1278
- build: update sphinx-autodoc-typehints version #1280
- chore: update SECURITY.md #1286

11.5.2 Fixed

- fix: manifest path should render as posix rather than uri #1289

11.5.3 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

11.5.4 Miscellaneous

- chore: pre-commit automatic update #1281, #1288

11.6 0.6.4 - 2024-06-24

11.6.1 Added

- feat: using ruff as the main linter/formatter #1274

11.6.2 Changed

- chore: update CHANGELOG for v0.6.3 #1273
- chore: bump pre-commit-hook version #1276

11.6.3 Fixed

- fix: backticks breaking doc build after ruff linter #1275

11.6.4 Dependencies

- build: bump pint from 0.24 to 0.24.1 #1277

11.7 0.6.3 - 2024-06-18

11.7.1 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

11.7.2 Fixed

- fix: unnecessary Point3D comparison #1264
- docs: examples are not being uploaded as assets (.py/.ipynb) #1268
- fix: change action order #1269

11.7.3 Dependencies

- build: bump numpy from 1.26.4 to 2.0.0 #1266
- build: bump the docs-deps group with 2 updates #1271

11.7.4 Miscellaneous

- chore: pre-commit automatic update #1267

11.8 0.6.2 - 2024-06-17

11.8.1 Added

- feat: deprecating log_level and logs_folder + adding client log control #1260
- feat: adding deprecation support for args and methods #1261

11.8.2 Changed

- chore: update CHANGELOG for v0.6.1 #1256
- ci: simplify doc build using ansys/actions #1262

11.8.3 Fixed

- fix: Rename built in shadowing variables #1257

11.9 0.6.1 - 2024-06-12

11.9.1 Added

- feat: revolve a sketch given an axis and an origin #1248

11.9.2 Changed

- chore: update CHANGELOG for v0.6.0 #1245
- chore: update dev version to 0.8.dev0 #1246

11.9.3 Fixed

- fix: Bug in *show* function #1255

11.9.4 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

11.9.5 Miscellaneous

- docs: add conda information for package #1247

11.10 0.6.0 - 2024-06-07

11.10.1 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

11.10.2 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

11.10.3 Fixed

- fix: Interactive documentation #1226
- fix: only notify on failure and fill with data #1238

11.10.4 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

11.10.5 Miscellaneous

- docs: update AUTHORS #1222

11.11 0.5.6 - 2024-05-23

11.11.1 Added

- feat: add new arc constructors #1208

11.11.2 Changed

- chore: update CHANGELOG for v0.5.5 #1205

11.11.3 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

11.12 0.5.5 - 2024-05-21

11.12.1 Changed

- docs: adapt `ansys_sphinx_theme_autoapi` extension for `autoapi` #1135
- chore: update CHANGELOG for v0.5.4 #1194

11.12.2 Fixed

- fix: adapting `Arc` class constructor order to (start, end, center) #1196
- chore: limit requests library version under 2.32 #1203

11.12.3 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

11.12.4 Miscellaneous

- chore: pre-commit automatic update #1202

11.13 0.5.4 - 2024-05-15

11.13.1 Added

- feat: allow for `product_version` on geometry service launcher function #1182

11.13.2 Changed

- chore: update CHANGELOG for v0.5.3 #1177

11.13.3 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

11.13.4 Miscellaneous

- chore: pre-commit automatic update #1180, #1193
- docs: add geometry preparation for Fluent simulation #1183

11.14 0.5.3 - 2024-04-29

11.14.1 Fixed

- fix: semver intersphinx mapping not resolved properly #1175
- fix: start and end points for edge #1176

11.15 0.5.2 - 2024-04-29

11.15.1 Added

- feat: add semver to intersphinx #1173

11.15.2 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

11.15.3 Fixed

- fix: GetSurface and GetCurve not available prior to 24R2 #1171

11.15.4 Miscellaneous

- docs: creating a NACA airfoil example #1167
- docs: simplify README example #1169

11.16 0.5.1 - 2024-04-24

11.16.1 Added

- feat: security updates dropped for v0.3 or earlier #1126
- feat: add `export_to` functions #1147

11.16.2 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

11.16.3 Fixed

- fix: docs link in example #1137
- fix: update backend version message #1145
- fix: Trame issues #1148
- fix: Interactive documentation #1160

11.16.4 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

11.16.5 Miscellaneous

- docs: add example on exporting designs #1149
- docs: fix link in *CHANGELOG.md* #1154
- chore: pre-commit automatic update #1159

11.17 0.5.0 - 2024-04-17

11.17.1 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
- feat: sweeping chains and profiles #1056
- feat: vulnerability checks #1071
- feat: loft profiles #1075
- feat: accept bandit advisories in-line for subprocess #1077
- feat: adding containers to automatic launcher #1090
- feat: minor changes to Linux Dockerfile #1111
- feat: avoid error if folder exists #1125

11.17.2 Changed

- build: changing sphinx-autoapi from 3.1.a2 to 3.1.a4 #1038
- chore: add pre-commit.ci configuration #1065
- chore: dependabot PR automatic approval #1067
- ci: bump the actions group with 1 update #1082
- chore: update docker tags to be kept #1085
- chore: update pre-commit versions #1094
- build: use ansys-sphinx-theme autoapi target #1097
- fix: removing @PipKat from `*.md` files - changelog fragments #1098

- ci: dashboard upload does not apply anymore #1099
- chore: pre-commit.ci not working properly #1108
- chore: update and adding pre-commit.ci config hook #1109
- ci: main Python version update to 3.12 #1112
- ci: skip Linux tests with common approach #1113
- ci: build changelog on release #1118
- chore: update CHANGELOG for v0.5.0 #1119

11.17.3 Fixed

- feat: re-enable open file on Linux #817
- fix: adapt export and download tests to new hoops #1057
- fix: linux Dockerfile - replace .NET6.0 references by .NET8.0 #1069
- fix: misleading docstring for sweep_chain() #1070
- fix: prepare_and_start_backend is only available on Windows #1076
- fix: unit tests failing after dms update #1087
- build: beartype upper limit on v0.18 #1095
- fix: improper types being passed for Face and Edge ctor. #1096
- fix: return type should be dict and not `ScalarMapContainer` (grpc type) #1103
- fix: env version for Dockerfile Windows #1120
- fix: changelog description ill-formatted #1121
- fix: solve issues with intersphinx when releasing #1123

11.17.4 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
- build: bump pytest-cov from 4.1.0 to 5.0.0 #1081
- build: bump ansys-api-geometry from 0.4.0 to 0.4.1 #1092
- build: bump beartype from 0.17.2 to 0.18.2 #1106
- build: bump ansys-tools-path from 0.4.1 to 0.5.1 #1107
- build: bump panel from 1.4.0 to 1.4.1 in the docs-deps group #1114
- build: bump scipy from 1.12.0 to 1.13.0 #1115

11.17.5 Miscellaneous

- [pre-commit.ci] pre-commit autoupdate #1063
- docs: add examples on new methods #1089
- chore: pre-commit automatic update #1116

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