PyAnsys Geometry

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Jul 15, 2025

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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 0.11.0.

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

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

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

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

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

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

Assets

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

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.

Note

Only Ansys employees with access to https://github.com/ansys/pyansys-geometry-binaries can download these binaries.

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 Ansys employees 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:

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

Here are some terms to keep in mind:

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

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

Using PyAnsys Geometry launcher

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

Linux/Mac

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

Powershell

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

Windows CMD

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

Warning

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

Manual launch

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

Geometry service launcher

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

Using PyAnsys Geometry launcher

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

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

modeler = launch_modeler()
```

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

Manual launch

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

Linux/Mac

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

Powershell

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

Windows CMD

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

Warning

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

Connect to the Geometry service

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

```
from ansys.geometry.core import Modeler

modeler = Modeler()
```

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

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

Warning

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

Environment variables

Linux/Mac

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

Powershell

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

Windows CMD

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

Modeler function

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

Go to Docker containers

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5.2 Launch a local session

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

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

Discovery

```
from ansys.geometry.core import launch_modeler_with_discovery

modeler = launch_modeler_with_discovery()
```

SpaceClaim

```
from ansys.geometry.core import launch_modeler_with_spaceclaim

modeler = launch_modeler_with_spaceclaim()
```

Geometry service

```
from ansys.geometry.core import launch_modeler_with_geometry_service

modeler = launch_modeler_with_geometry_service()
```

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

Powershell

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

Windows CMD

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

Linux

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

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

- launch_modeler_with_discovery
- launch_modeler_with_spaceclaim
- launch_modeler_with_geometry_service

Note

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

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5.3 Launch a remote session

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

Warning

This option is only available for Ansys employees.

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

5.3.1 Set up the client machine

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

   ```
   from ansys.discovery.core import launch_modeler_with_pimlight_and_discovery

   disco = launch_modeler_with_pimlight_and_discovery("241")
   ```

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

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

   ```
   from ansys.discovery.core import launch_modeler_with_pimlight_and_spaceclaim

   sc = launch_modeler_with_pimlight_and_spaceclaim("version")

   from ansys.discovery.core import launch_modeler_with_pimlight_and_geometry_service

   geo = launch_modeler_with_pimlight_and_geometry_service("version")
   ```

Note

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

5.3.2 End the session

To end the session, run the corresponding command:

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

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5.4 Use an existing session

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

Warning

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

5.4.1 Establish the connection

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

```
from ansys.geometry.core import Modeler

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

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

5.4.2 Verify the connection

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

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

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5.5 Ansys version compatibility

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

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

Tip

Forth- and back-compatibility mechanism

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

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

Access to the documentation for the preceding versions is found at the Versions page.

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5.6 Install package in development mode

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

Contents

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 3.13. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

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

```
unzip ansys-geometry-core-v0.11.0-all-wheelhouse-ubuntu-3.10.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

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

Consider installing using a virtual environment. For more information, see Creation of virtual environments in the Python documentation.

5.6.6 Verify your installation

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

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

Ansys Geometry Modeler (0x205c5c17d90)

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

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

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

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

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

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

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

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

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

Note

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

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

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

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

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

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

Note

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

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

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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"
```

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```
)
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))
```

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

6.3.4 Create bodies by extruding the sketch

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

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

6.3.5 Create bodies by extruding the face

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

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

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

- *Body()*
- *Component()*

6.3.6 Download and save design

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

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

For more information, see the *Design* submodule.

6.4 PyAnsys Geometry overview

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

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

6.5 Simple interactive example

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

6.5.1 Start Geometry server instance

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

This code starts an instance of the Geometry service:

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

6.5.2 Create geometry model

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

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

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

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

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

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.
parameters	PyAnsys Geometry parameters 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	Deprecated constant. Use `DISABLE_ACTIVE_DESIGN_CHECK` instead.
DISABLE_ACTIVE_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.
USE_TRACKER_TO_UPDATE_DESIGN	Global constant for checking whether to use the tracker to update designs.

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
CORE_WINDOWS
CORE_LINUX
DISCOVERY_HEADLESS

Static methods

is_core_service	Determine whether the backend is CoreService based or not.
is_headless_service	Determine whether the backend is a headless service or not.
is_linux_service	Determine whether the backend is a Linux service or not.

Import detail

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

Attribute detail

BackendType.**DISCOVERY = 0**

BackendType.**SPACECLAIM = 1**

BackendType.**WINDOWS_SERVICE = 2**

BackendType.**LINUX_SERVICE = 3**

BackendType.**CORE_WINDOWS = 4**

BackendType.**CORE_LINUX = 5**

BackendType.**DISCOVERY_HEADLESS = 6**

Method detail

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

Determine whether the backend is CoreService based or not.

Parameters

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

Returns

bool
True if the backend is CoreService based, False otherwise.

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

Determine whether the backend is a headless service or not.

Parameters

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

Returns

bool
True if the backend is a headless service, False otherwise.

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

Determine whether the backend is a Linux service or not.

Parameters

backend_type
[BackendType] The backend type to check whether or not it's running on Linux.

Returns

bool
True if the backend is running on Linux, False otherwise.

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
V_251
V_252

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**

ApiVersions.**V_251 = 251**

ApiVersions.**V_252 = 252**

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 = pygeom_defaults.DEFAULT_HOST*, *port: str | int = pygeom_defaults.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*, *proto_version: str | 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.

proto_version: str or None, default: None
Version of the gRPC protocol to use. If None, the latest version is used.

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.
services	GRPC services.
log	Specific instance logger.
is_closed	Flag indicating whether the client connection is closed.
healthy	Flag indicating whether the client channel is healthy.

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

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

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

Client gRPC channel.

property GrpcClient.**services: ansys.geometry.core._grpc._services._service._GRPCServices**

GRPC services.

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

Specific instance logger.

property GrpcClient.**is_closed: bool**

Flag indicating whether the client connection is closed.

property GrpcClient.**healthy: bool**

Flag indicating whether the client channel is healthy.

Method detail

GrpcClient.**__repr__**() → str

Represent the client as a string.

GrpcClient.**close**()

Close the channel.

Notes

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

GrpcClient.**target**() → str

Get the target of the channel.

GrpcClient.**get_name**() → str

Get the target name of the connection.

Description

Module providing a wrapped abstraction of the gRPC stubs.

Module detail

client.**wait_until_healthy**(*channel: grpc.Channel | str*, *timeout: float*) → grpc.Channel

Wait until a channel is healthy before returning.

Parameters

channel
[Channel | str] Channel that must be established and healthy. The target can also be passed in. In that case, a channel is created using the default insecure channel.

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

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

Returns

grpc.Channel
The channel that was passed in. This channel is guaranteed to be healthy. If a string was passed in, a channel is created using the default insecure channel.

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.
grpc_point_to_point3d	Convert a point gRPC message class to a `Point3D` class.
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.
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_curve	Convert a `TrimmedCurve` to a trimmed curve gRPC message.
line_to_grpc_line	Convert a `Line` to a line gRPC message.
grpc_material_to_material	Convert a material gRPC message to a `Material` class.
grpc_material_property_to_material_pro	Convert a material property gRPC message to a `MaterialProperty` class.

Description

Module providing for conversions.

Module detail

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

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

Parameters

unit_vector
[`UnitVector3D`] Source vector data.

Returns

GRPCDirection
Geometry service gRPC direction message.

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

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

Parameters

frame
[`Frame`] Source frame data.

Returns

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

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

Convert a `Plane` class to a plane gRPC message.

Parameters

plane
[`Plane`] Source plane data.

Returns

GRPCPlane
Geometry service gRPC plane message. The unit is meters.

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

Convert lists of `SketchEdge` and `SketchFace` to a gRPC message.

Parameters

plane
[`Plane`] Plane for positioning the 2D sketches.

edges
[`list`[`SketchEdge`]] Source edge data.

faces
[`list`[`SketchFace`]] Source face data.

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

Returns

GRPCGeometries
Geometry service gRPC geometries message. The unit is meters.

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

Convert a list of `SketchEdge` to a gRPC message.

Parameters

edges
[`list`[`SketchEdge`]] Source edge data.

plane
[`Plane`] Plane for positioning the 2D sketches.

Returns

tuple[list[GRPCLine], list[GRPCArc]]
Geometry service gRPC line and arc messages. The unit is meters.

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

Convert an `Arc` class to an arc gRPC message.

Parameters

arc
[`Arc`] Source arc data.

plane
[`Plane`] Plane for positioning the arc within.

Returns

GRPCArc
Geometry service gRPC arc message. The unit is meters.

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

Convert a `SketchEllipse` class to an ellipse gRPC message.

Parameters

ellipse
[`SketchEllipse`] Source ellipse data.

Returns

GRPCEllipse
Geometry service gRPC ellipse message. The unit is meters.

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

Convert a `SketchCircle` class to a circle gRPC message.

Parameters

circle
[`SketchCircle`] Source circle data.

plane
[`Plane`] Plane for positioning the circle.

Returns

GRPCCircle
Geometry service gRPC circle message. The unit is meters.

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

Convert a `Point3D` class to a point gRPC message.

Parameters

point
[`Point3D`] Source point data.

Returns

GRPCPoint
Geometry service gRPC point message. The unit is meters.

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

Convert a point gRPC message class to a `Point3D` class.

Parameters

point
[`GRPCPoint`] Source point data.

Returns

Point3D
Converted point.

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

Convert a `Point2D` class to a point gRPC message.

Parameters

plane
[`Plane`] Plane for positioning the 2D point.

point
[`Point2D`] Source point data.

Returns

GRPCPoint
Geometry service gRPC point message. The unit is meters.

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

Convert a `Polygon` class to a polygon gRPC message.

Parameters

polygon
[`Polygon`] Source polygon data.

Returns

GRPCPolygon
Geometry service gRPC polygon message. The unit is meters.

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

Convert a `Segment` class to a line gRPC message.

Parameters

segment
[`SketchSegment`] Source segment data.

Returns

GRPCLine
Geometry service gRPC line message. The unit is meters.

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

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

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

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

Parameters

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

Returns

frame
[`Frame`] Resulting converted frame.

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

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

Parameters

surface
[`GRPCSurface`] Geometry service gRPC surface message.

Returns

Surface
Resulting converted surface.

conversions.**grpc_curve_to_curve**(*curve: ansys.api.geometry.v0.models_pb2.CurveGeometry*) → *ansys.geometry.core.shapes.curves.curve.Curve*

Convert a curve gRPC message to a `Curve`.

Parameters

curve
[`GRPCCurve`] Geometry service gRPC curve message.

Returns

Curve
Resulting converted curve.

conversions.**curve_to_grpc_curve**(*curve:* ansys.geometry.core.shapes.curves.curve.Curve) → ansys.api.geometry.v0.models_pb2.CurveGeometry

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

Parameters

curve
[`Curve`] Curve to convert.

Returns

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

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

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

Parameters

curve
[`TrimmedCurve`] Curve to convert.

Returns

GRPCTrimmedCurve
Geometry service gRPC `TrimmedCurve` message.

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

Convert a `Line` to a line gRPC message.

Parameters

line
[`Line`] Line to convert.

Returns

GRPCLine
Geometry service gRPC `Line` message.

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

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

Parameters

material
[`GRPCMaterial`] Material gRPC message.

Returns

Material
Converted material.

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

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

Parameters

material_property
[`GRPCMaterialProperty`] Material property gRPC message.

Returns

MaterialProperty
Converted material property.

The `defaults.py` module

Summary

Constants

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 = pygeom_defaults.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

CORE_WINDOWS_LATEST
CORE_LINUX_LATEST
CORE_WINDOWS_LATEST_UNSTABLE
CORE_LINUX_LATEST_UNSTABLE
WINDOWS_LATEST
WINDOWS_LATEST_UNSTABLE
WINDOWS_24_1
WINDOWS_24_2
WINDOWS_25_1
WINDOWS_25_2
CORE_WINDOWS_25_2
CORE_LINUX_25_2
CORE_WINDOWS_26_1
CORE_LINUX_26_1

Import detail

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

Attribute detail

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

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

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

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

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

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

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

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

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

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

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

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

GeometryContainers.**CORE_WINDOWS_26_1 = (10, 'windows', 'core-windows-26.1')**

GeometryContainers.**CORE_LINUX_26_1 = (11, 'linux', 'core-linux-26.1')**

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_pimlig	Start Ansys Discovery remotely using the PIM API.
launch_modeler_with_geometry_service_and	Start the Geometry service remotely using the PIM API.
launch_modeler_with_spaceclaim_and_pimli	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.
launch_modeler_with_core_service	Start the Geometry Core service locally using the `ProductInstance` class.

Description

Module for connecting to instances of the Geometry service.

Module detail

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

Start the `Modeler` interface for PyAnsys Geometry.

Parameters

mode

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

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

****kwargs**

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

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

Returns

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

Examples

Launch the Geometry service.

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

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

Start the Geometry service remotely using the PIM API.

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

Parameters

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

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

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

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

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

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

Returns

ansys.geometry.core.modeler.Modeler
Instance of the Geometry service.

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

Start the Geometry service locally using Docker.

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

Parameters

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

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

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

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

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

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

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

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

Returns

Modeler
Instance of the Geometry service.

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

Start Ansys Discovery remotely using the PIM API.

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

Parameters

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

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

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

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

Returns

ansys.geometry.core.modeler.Modeler
Instance of Modeler.

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

Start the Geometry service remotely using the PIM API.

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

Parameters

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

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

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

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

Returns

ansys.geometry.core.modeler.Modeler
Instance of Modeler.

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

Start Ansys SpaceClaim remotely using the PIM API.

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

Parameters

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

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

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

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

Returns

ansys.geometry.core.modeler.Modeler
Instance of Modeler.

launcher.**launch_modeler_with_geometry_service**(*version: str | int | None = 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*, *product_version: int = 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

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

Ansys products versions and their corresponding int values:

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

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

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

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

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

server_log_level
[int, optional]

Backend's log level from 0 to 3:
0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

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

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

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

product_version: int, optional
The product version to be started. Deprecated, use *version* instead.

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

Returns

Modeler
Instance of the Geometry service.

Raises

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

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

Examples

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

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

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

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

launcher.**launch_modeler_with_discovery**(*version: str | int | None = 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*, *product_version: int = None*, ***kwargs: dict | None*)

Start Ansys Discovery locally using the `ProductInstance` class.

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

Parameters

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

Ansys products versions and their corresponding int values:

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

host: str, optional
IP address at which the Discovery session will be deployed. By default, its value will be `localhost`.

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

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

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

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

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

server_log_level
[`int`, `optional`]

Backend's log level from 0 to 3:
0: Chatterbox 1: Debug 2: Warning 3: Error

The default is `2` (Warning).

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

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

product_version: int, optional
The product version to be started. Deprecated, use *version* instead.

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

Returns

Modeler
Instance of the Geometry service.

Raises

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

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

Examples

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

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

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

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

launcher.**launch_modeler_with_spaceclaim**(*version: str | int | None = 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*, *product_version: 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

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

Ansys products versions and their corresponding int values:

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

host: str, optional
IP address at which the SpaceClaim session will be deployed. By default, its value will be `localhost`.

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

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

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

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

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

server_log_level
[int, optional]

Backend's log level from 0 to 3:
0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

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

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

product_version: int, optional
The product version to be started. Deprecated, use *version* instead.

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

Returns

Modeler
Instance of the Geometry service.

Raises

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

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

Examples

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

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

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

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

launcher.**launch_modeler_with_core_service**(*version: str | int | None = 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*, *product_version: int = None*, ***kwargs: dict | None*) → *ansys.geometry.core.modeler.Modeler*

Start the Geometry Core service locally using the `ProductInstance` class.

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

Parameters

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

Ansys products versions and their corresponding int values:

- `252` : Ansys 25R2
- `261` : Ansys 26R1

host: str, optional
IP address at which the service will be deployed. By default, its value will be `localhost`.

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

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

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

server_log_level
[`int`, `optional`]

Backend's log level from 0 to 3:
0: Chatterbox 1: Debug 2: Warning 3: Error

The default is `2` (Warning).

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

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

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

product_version: int, optional
The product version to be started. Deprecated, use *version* 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 Core service.

Raises

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

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

Examples

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

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

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

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

The `product_instance.py` module

Summary

Classes

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_FOLDE	Default Geometry Service's folder name into the unified installer (DMS).
CORE_GEOMETRY_SERVICE_FOLDER	Default Geometry Service's folder name into the unified installer (Core Service).
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 (DMS).
CORE_GEOMETRY_SERVICE_EXE	The Windows Geometry Service's filename (Core Service).
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_ARGUMEN	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_ENVV	SpaceClaim hidden backend's environment variable key.
BACKEND_SPACECLAIM_HIDDEN_ENVV	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.
ANSYS_GEOMETRY_SERVICE_ROOT	Local Geometry Service install location. This is for GeometryService and CoreGeometryService.

ProductInstance

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

`ProductInstance` class.

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

Parameters

pid

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

Overview

Methods

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, *version: str | int = None*, *host: str = 'localhost'*, *port: int = None*, *enable_trace: bool = False*, *api_version:* ansys.geometry.core.connection.backend.ApiVersions = *ApiVersions.LATEST*, *timeout: int = 150*, *manifest_path: str = None*, *hidden: bool = False*, *server_log_level: int = 2*, *client_log_level: int = logging.INFO*, *server_logs_folder: str = None*, *client_log_file: str = None*, *specific_minimum_version: int = None*, *product_version: int | None = 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

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

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

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

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

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

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

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

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

server_log_level
[int, optional]

Backend's log level from 0 to 3:
0: Chatterbox 1: Debug 2: Warning 3: Error

The default is 2 (Warning).

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

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

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

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

product_version: ``int``, optional
The product version to be started. Deprecated, use *version* instead.

Returns

Modeler
Instance of the Geometry service.

Raises

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

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

product_instance.**get_available_port**() → int

Return an available port to be used.

Returns

int
The available port.

product_instance.**WINDOWS_GEOMETRY_SERVICE_FOLDER = 'GeometryService'**
Default Geometry Service's folder name into the unified installer (DMS).

product_instance.**CORE_GEOMETRY_SERVICE_FOLDER = 'GeometryService'**
Default Geometry Service's folder name into the unified installer (Core Service).

product_instance.**DISCOVERY_FOLDER = 'Discovery'**
Default Discovery's folder name into the unified installer.

product_instance.**SPACECLAIM_FOLDER = 'scdm'**
Default SpaceClaim's folder name into the unified installer.

product_instance.**ADDINS_SUBFOLDER = 'Addins'**
Default global Addins's folder name into the unified installer.

product_instance.**BACKEND_SUBFOLDER = 'ApiServer'**
Default backend's folder name into the ADDINS_SUBFOLDER folder.

product_instance.**MANIFEST_FILENAME = 'Presentation.ApiServerAddIn.Manifest.xml'**
Default backend's add-in filename.

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

product_instance.**GEOMETRY_SERVICE_EXE = 'Presentation.ApiServerDMS.exe'**
The Windows Geometry Service's filename (DMS).

product_instance.**CORE_GEOMETRY_SERVICE_EXE = 'Presentation.ApiServerCoreService.exe'**
The Windows Geometry Service's filename (Core Service).

product_instance.**DISCOVERY_EXE = 'Discovery.exe'**
The Ansys Discovery's filename.

product_instance.**SPACECLAIM_EXE = 'SpaceClaim.exe'**
The Ansys SpaceClaim's filename.

product_instance.**BACKEND_LOG_LEVEL_VARIABLE = 'LOG_LEVEL'**
The backend's log level environment variable for local start.

product_instance.**BACKEND_TRACE_VARIABLE = 'ENABLE_TRACE'**
The backend's enable trace environment variable for local start.

product_instance.**BACKEND_HOST_VARIABLE = 'API_ADDRESS'**
The backend's ip address environment variable for local start.

product_instance.**BACKEND_PORT_VARIABLE = 'API_PORT'**
The backend's port number environment variable for local start.

product_instance.**BACKEND_LOGS_FOLDER_VARIABLE = 'ANS_DSCO_REMOTE_LOGS_FOLDER'**
The backend's logs folder path to be used.

Only applicable to the Ansys Geometry Service.

product_instance.**BACKEND_API_VERSION_VARIABLE = 'API_VERSION'**

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

To be used only with Ansys Discovery and Ansys SpaceClaim.

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

The additional argument for local Ansys Discovery start.

To be used only with Ansys Discovery.

product_instance.**BACKEND_ADDIN_MANIFEST_ARGUMENT = '/ADDINMANIFESTFILE='**

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

To be used only with Ansys Discovery and Ansys SpaceClaim.

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

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

To be used only with Ansys SpaceClaim.

product_instance.**BACKEND_SPACECLAIM_HIDDEN_ENVVAR_KEY = 'SPACECLAIM_MODE'**

SpaceClaim hidden backend's environment variable key.

To be used only with Ansys SpaceClaim.

product_instance.**BACKEND_SPACECLAIM_HIDDEN_ENVVAR_VALUE = '2'**

SpaceClaim hidden backend's environment variable value.

To be used only with Ansys SpaceClaim.

product_instance.**BACKEND_DISCOVERY_HIDDEN = '--hidden'**

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

To be used only with Ansys Discovery.

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

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

To be used only with Ansys Discovery and Ansys SpaceClaim.

product_instance.**ANSYS_GEOMETRY_SERVICE_ROOT = 'ANSYS_GEOMETRY_SERVICE_ROOT'**

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

The `validate.py` module

Summary

Functions

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.
geometry_commands	Provides tools for pulling geometry.
mating_conditions	Provides dataclasses for mating conditions.
part	Module providing fundamental data of an assembly.
selection	Module for creating a named selection.
vertex	Module for managing a vertex.

The `beam.py` module

Summary

Classes

BeamProfile	Represents a single beam profile organized within the design assembly.
BeamCircularProfile	Represents a single circular beam profile.
BeamCrossSectionInfo	Represents the cross-section information for a beam.
BeamProperties	Represents the properties of a beam.
Beam	Represents a simplified solid body with an assigned 2D cross-section.

Enums

BeamType	Provides values for the beam types supported.
SectionAnchorType	Provides values for the section anchor types supported.

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

```
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

```
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.

BeamCrossSectionInfo

class ansys.geometry.core.designer.beam.**BeamCrossSectionInfo**(*section_anchor:* SectionAnchorType, *section_angle:* *float*, *section_frame:* ansys.geometry.core.math.frame.Frame, *section_profile:* *list*[*list*[ansys.geometry.core.shapes.curves.trimmed_cu | *None*)

Represents the cross-section information for a beam.

Parameters

section_anchor
[`SectionAnchorType`] Specifies how the beam section is anchored to the beam path.

section_angle
[`float`] The rotation angle of the cross section clockwise from the default perpendicular of the beam path.

section_frame
[`Frame`] The section frame at the start of the beam.

section_profile
[`BeamProfile`] The section profile in the XY plane.

Overview

Properties

section_anchor	Specifies how the beam section is anchored to the beam path.
section_angle	Rotation angle of the cross section clockwise from the perpendicular of the beam path.
section_frame	The section frame at the start of the beam.
section_profile	The section profile in the XY plane.

Special methods

__repr__	Represent the `BeamCrossSectionInfo` as a string.

Import detail

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

Property detail

property BeamCrossSectionInfo.**section_anchor: *SectionAnchorType***

Specifies how the beam section is anchored to the beam path.

property BeamCrossSectionInfo.**section_angle: float**

Rotation angle of the cross section clockwise from the perpendicular of the beam path.

property BeamCrossSectionInfo.**section_frame: *ansys.geometry.core.math.frame.Frame***

The section frame at the start of the beam.

property BeamCrossSectionInfo.**section_profile:**
list[list[*ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve*]] | None

The section profile in the XY plane.

Method detail

BeamCrossSectionInfo.**__repr__**() → str

Represent the BeamCrossSectionInfo as a string.

BeamProperties

class ansys.geometry.core.designer.beam.**BeamProperties**(*area: float*, *centroid:* ansys.geometry.core.shapes.parameterization.ParamUV, *warping_constant: float*, *ixx: float*, *ixy: float*, *iyy: float*, *shear_center:* ansys.geometry.core.shapes.parameterization.ParamUV, *torsion_constant: float*)

Represents the properties of a beam.

Parameters

area
[float] The cross-sectional area of the beam.

centroid
[ParamUV] The centroid of the beam section.

warping_constant
[float] The warping constant of the beam.

ixx
[float] The moment of inertia about the x-axis.

ixy
[float] The product of inertia.

iyy
[float] The moment of inertia about the y-axis.

shear_center
[ParamUV] The shear center of the beam.

torsion_constant
[float] The torsion constant of the beam.

Overview

Properties

area	The cross-sectional area of the beam.
centroid	The centroid of the beam section.
warping_constant	The warping constant of the beam.
ixx	The moment of inertia about the x-axis.
ixy	The product of inertia.
iyy	The moment of inertia about the y-axis.
shear_center	The shear center of the beam.
torsion_constant	The torsion constant of the beam.

Import detail

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

Property detail

property BeamProperties.**area: float**

The cross-sectional area of the beam.

property BeamProperties.**centroid:** *ansys.geometry.core.shapes.parameterization.ParamUV*

The centroid of the beam section.

property BeamProperties.**warping_constant: float**

The warping constant of the beam.

property BeamProperties.**ixx: float**

The moment of inertia about the x-axis.

property BeamProperties.**ixy: float**

The product of inertia.

property BeamProperties.**iyy: float**

The moment of inertia about the y-axis.

property BeamProperties.**shear_center:** *ansys.geometry.core.shapes.parameterization.ParamUV*

The shear center of the beam.

property BeamProperties.**torsion_constant: float**

The torsion constant of the beam.

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 | *None*, *parent_component:* ansys.geometry.core.designer.component.Component, *name: str = None*, *is_deleted: bool = False*, *is_reversed: bool = False*, *is_rigid: bool = False*, *material:* ansys.geometry.core.materials.material.Material *= None*, *cross_section:* BeamCrossSectionInfo *= None*, *properties:* BeamProperties *= None*, *shape:* ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve *= None*, *beam_type:* BeamType *= None*)

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.

BeamType

class ansys.geometry.core.designer.beam.**BeamType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for the beam types supported.

Overview

Attributes

BEAM
SPRING
LINK_TRUSS
CABLE
PIPE
THERMALFLUID
UNKNOWN

Import detail

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

Attribute detail

BeamType.**BEAM = 0**

BeamType.**SPRING = 1**

BeamType.**LINK_TRUSS = 2**

BeamType.**CABLE = 3**

BeamType.**PIPE = 4**

BeamType.**THERMALFLUID = 5**

BeamType.**UNKNOWN = 6**

SectionAnchorType

class ansys.geometry.core.designer.beam.**SectionAnchorType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for the section anchor types supported.

Overview

Attributes

CENTROID
SHEAR_CENTER
ANCHOR_LOCATION

Import detail

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

Attribute detail

SectionAnchorType.**CENTROID = 0**

SectionAnchorType.**SHEAR_CENTER = 1**

SectionAnchorType.**ANCHOR_LOCATION = 2**

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.

Constants

__TEMPORARY_BOOL_OPS_FIX__

IBody

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

Bases: abc.ABC

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

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

Overview

Abstract methods

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.
is_suppressed	Get the body suppression state.
set_suppressed	Set the body suppression state.
color	Get the color of the body.
opacity	Get the float value of the opacity for 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.
vertices	Get a list of all vertices 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.
material	Get the assigned material of the body.
bounding_box	Get the bounding box of the body.
assign_material	Assign a material against the active design.

continues on next page

Table 1 – continued from previous page

get_assigned_material	Get the assigned material of the body.
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.
shell_body	Shell the body to the thickness specified.
remove_faces	Shell by removing a given set of faces.
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

abstractmethod IBody.**id**() → str

Get the ID of the body as a string.

abstractmethod IBody.**name**() → str

Get the name of the body.

abstractmethod IBody.**set_name**(*str*) → None

Set the name of the body.

> **Warning**
>
> This method is only available starting on Ansys release 25R1.

abstractmethod IBody.**fill_style**() → *FillStyle*

Get the fill style of the body.

abstractmethod IBody.**set_fill_style**(*fill_style:* FillStyle) → None

Set the fill style of the body.

Warning

This method is only available starting on Ansys release 25R1.

abstractmethod IBody.**is_suppressed**() → bool

Get the body suppression state.

Warning

This method is only available starting on Ansys release 25R2.

abstractmethod IBody.**set_suppressed**(*suppressed: bool*) → None

Set the body suppression state.

Warning

This method is only available starting on Ansys release 25R2.

abstractmethod IBody.**color**() → str

Get the color of the body.

abstractmethod IBody.**opacity**() → float

Get the float value of the opacity for the body.

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

Set the color of the body.

Parameters

color
[str | tuple[float, float, float] | tuple[float, 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).

Warning

This method is only available starting on Ansys release 25R1.

abstractmethod IBody.**faces**() → list[*ansys.geometry.core.designer.face.Face*]

Get a list of all faces within the body.

Returns

list[Face]

abstractmethod IBody.**edges**() → list[*ansys.geometry.core.designer.edge.Edge*]

Get a list of all edges within the body.

Returns

list[Edge]

abstractmethod IBody.**vertices**() → list[*ansys.geometry.core.designer.vertex.Vertex*]

Get a list of all vertices within the body.

Returns

list[**Vertex**]

abstractmethod IBody.**is_alive**() → bool

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

abstractmethod IBody.**is_surface**() → bool

Check if the body is a planar body.

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

abstractmethod IBody.**material**() → *ansys.geometry.core.materials.material.Material*

Get the assigned material of the body.

Returns

Material
Material assigned to the body.

abstractmethod IBody.**bounding_box**() → *ansys.geometry.core.math.bbox.BoundingBox*

Get the bounding box of the body.

Returns

BoundingBox
Bounding box of the body.

Warning

This method is only available starting on Ansys release 25R2.

abstractmethod IBody.**assign_material**(*material:* ansys.geometry.core.materials.material.Material) → None

Assign a material against the active design.

Parameters

material
[Material] Source material data.

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

Get the assigned material of the body.

Returns

Material
Material assigned to the body.

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

Warning

This method is only available starting on Ansys release 24R2.

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

Scale the geometry body by the given value.

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

Parameters

value: Real
Value to scale the body by.

Warning

This method is only available starting on Ansys release 24R2.

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

Map the geometry body to the new specified frame.

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

Parameters

frame: Frame
Structure defining the orientation of the body.

Warning

This method is only available starting on Ansys release 24R2.

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

Mirror the geometry body across the specified plane.

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

Parameters

plane: Plane
Represents the mirror.

Warning

This method is only available starting on Ansys release 24R2.

abstractmethod 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.

Warning

This method is only available starting on Ansys release 24R2.

abstractmethod 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.

abstractmethod IBody.**tessellate**(*merge: bool = False, tess_options:* ansys.geometry.core.misc.options.TessellationOptions | *None = None*) → 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.

tess_options
[`TessellationOptions` | `None`, default: `None`] A set of options to determine the tessellation quality.

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
```

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

Shell the body to the thickness specified.

Parameters

offset
[Real] Shell thickness.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

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

Shell by removing a given set of faces.

Parameters

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

offset
[Real] Shell thickness.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

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

Plot the body.

Parameters

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

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

use_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.
get_assigned_material	Get the assigned material of the body.
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_suppressed	Set the body suppression state.
set_color	Set the color of the body.
set_opacity	Set the opacity 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.
shell_body	Shell the body to the thickness specified.
remove_faces	Shell by removing a given set of faces.

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.
is_suppressed	Get the body suppression state.
color	Get the current color of the body.
opacity	Get the float value of the opacity for 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.
vertices	Get a list of all vertices within the body.
is_alive	Check if the body is still alive and has not been deleted.
volume	Calculate the volume of the body.
material	Get the assigned material of the body.
bounding_box	Get the bounding box 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.**is_suppressed: bool**

Get the body suppression state.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

property MasterBody.**color: str**

Get the current color of the body.

property MasterBody.**opacity: float**

Get the float value of the opacity for 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.**vertices**: list[*ansys.geometry.core.designer.vertex.Vertex*]

Get a list of all vertices within the body.

Returns

list[Vertex]

property MasterBody.**is_alive**: bool

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

property MasterBody.**volume**: pint.Quantity

Calculate the volume of the body.

Notes

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

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

Get the assigned material of the body.

Returns

Material
Material assigned to the body.

property MasterBody.**bounding_box**: *ansys.geometry.core.math.bbox.BoundingBox*

Get the bounding box of the body.

Returns

BoundingBox
Bounding box of the body.

Warning

This method is only available starting on Ansys release 25R2.

Method detail

`MasterBody.reset_tessellation_cache()`

Decorate `MasterBody` methods that need tessellation cache update.

Parameters

func
[`method`] Method to call.

Returns

Any
Output of the method, if any.

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

Assign a material against the active design.

Parameters

material
[`Material`] Source material data.

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

Get the assigned material of the body.

Returns

Material
Material assigned to the body.

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

Add a mid-surface thickness to a surface body.

Parameters

thickness
[`Quantity`] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters

offset_type
[`MidSurfaceOffsetType`] Surface offset to assign.

Notes

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

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.

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

Set the name of the body.

Warning

This method is only available starting on Ansys release 25R1.

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

Set the fill style of the body.

Warning

This method is only available starting on Ansys release 25R1.

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

Set the body suppression state.

Warning

This method is only available starting on Ansys release 25R2.

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

Set the color of the body.

Parameters

color
[str | tuple[float, float, float] | tuple[float, 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).

Warning

This method is only available starting on Ansys release 25R1.

MasterBody.**set_opacity**(*opacity: float*) → None

Set the opacity of the body.

Warning

This method is only available starting on Ansys release 25R2.

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.

Warning

This method is only available starting on Ansys release 24R2.

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

Scale the geometry body by the given value.

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

Parameters

value: Real
Value to scale the body by.

Warning

This method is only available starting on Ansys release 24R2.

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

Map the geometry body to the new specified frame.

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

Parameters

frame: Frame
Structure defining the orientation of the body.

Warning

This method is only available starting on Ansys release 24R2.

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

Mirror the geometry body across the specified plane.

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

Parameters

plane: Plane
Represents the mirror.

Warning

This method is only available starting on Ansys release 24R2.

`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.

Warning

This method is only available starting on Ansys release 24R2.

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*, *tess_options:* ansys.geometry.core.misc.options.TessellationOptions | *None = None*) → 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.

tess_options
[`TessellationOptions` | `None`, default: `None`] A set of options to determine the tessellation quality.

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
```

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```
     Y Bounds:  -1.000, 0.000
     Z Bounds:  -0.500, 4.500
```

Merge the body:

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

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

Shell the body to the thickness specified.

Parameters

offset
[Real] Shell thickness.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

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

Shell by removing a given set of faces.

Parameters

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

offset
[Real] Shell thickness.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

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

Plot the body.

Parameters

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

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

use_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)
```

abstractmethod 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.

abstractmethod 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.

abstractmethod 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.
get_assigned_material	Get the assigned material of the body.
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 curves onto the specified faces using a sketch or edges.
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_suppressed	Set the body suppression state.
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.
shell_body	Shell the body to the thickness specified.
remove_faces	Shell by removing a given set of faces.
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.
is_suppressed	Get the body suppression state.
color	Get the color of the body.
opacity	Get the float value of the opacity for the body.
parent_component	
faces	Get a list of all faces within the body.
edges	Get a list of all edges within the body.
vertices	Get a list of all vertices 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.
material	Get the assigned material of the body.
bounding_box	Get the bounding box 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.**is_suppressed: bool**

Get the body suppression state.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

property Body.**color: str**

Get the color of the body.

property Body.opacity: float

Get the float value of the opacity for 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.vertices: list[*ansys.geometry.core.designer.vertex.Vertex*]

Get a list of all vertices within the body.

Returns

list[Vertex]

property Body.is_alive: bool

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

property Body.is_surface: bool

Check if the body is a planar body.

property Body.surface_thickness: pint.Quantity | None

Get the surface thickness of a surface body.

Notes

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

property Body.surface_offset: *MidSurfaceOffsetType* | None

Get the surface offset type of a surface body.

Notes

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

property Body.volume: pint.Quantity

Calculate the volume of the body.

Notes

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

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

Get the assigned material of the body.

Returns

Material
Material assigned to the body.

property Body.**bounding_box: *ansys.geometry.core.math.bbox.BoundingBox***

Get the bounding box of the body.

Returns

BoundingBox
Bounding box of the body.

Warning

This method is only available starting on Ansys release 25R2.

Method detail

Body.**reset_tessellation_cache**()

Decorate Body methods that require a tessellation cache update.

Parameters

func
[method] Method to call.

Returns

Any
Output of the method, if any.

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

Assign a material against the active design.

Parameters

material
[Material] Source material data.

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

Get the assigned material of the body.

Returns

Material
Material assigned to the body.

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

Add a mid-surface thickness to a surface body.

Parameters

thickness
[Quantity] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters

offset_type
[MidSurfaceOffsetType] Surface offset to assign.

Notes

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

Body.**imprint_curves**(*faces: list[*ansys.geometry.core.designer.face.Face*]*, *sketch:* ansys.geometry.core.sketch.sketch.Sketch = *None*, *trimmed_curves: list[*ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve*] = None*) → tuple[list[*ansys.geometry.core.designer.edge.Edge*], list[*ansys.geometry.core.designer.face.Face*]]

Imprint curves onto the specified faces using a sketch or edges.

Parameters

faces
[list[Face]] The list of faces to imprint the curves onto.

sketch
[Sketch, optional] The sketch containing curves to imprint.

trimmed_curves
[list[TrimmedCurve], optional] The list of curves to be imprinted. If sketch is provided, this parameter is ignored.

Returns

tuple[list[Edge], list[Face]]
A tuple containing the list of new edges and faces created by 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.

> **Warning**
>
> This method is only available starting on Ansys release 25R1.

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

Set the fill style of the body.

> **Warning**
>
> This method is only available starting on Ansys release 25R1.

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

Set the body suppression state.

Warning

This method is only available starting on Ansys release 25R2.

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

Set the color of the body.

Parameters

color
[str | tuple[float, float, float] | tuple[float, 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).

Warning

This method is only available starting on Ansys release 25R1.

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.

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.

Warning

This method is only available starting on Ansys release 24R2.

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

Scale the geometry body by the given value.

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

Parameters

value: Real
Value to scale the body by.

Warning

This method is only available starting on Ansys release 24R2.

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

Map the geometry body to the new specified frame.

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

Parameters

frame: Frame
Structure defining the orientation of the body.

Warning

This method is only available starting on Ansys release 24R2.

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

Mirror the geometry body across the specified plane.

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

Parameters

plane: Plane
Represents the mirror.

Warning

This method is only available starting on Ansys release 24R2.

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.

Warning

This method is only available starting on Ansys release 24R2.

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*, *tess_options:* ansys.geometry.core.misc.options.TessellationOptions | *None = None*) → 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.

tess_options
[`TessellationOptions` | `None`, default: `None`] A set of options to determine the tessellation quality.

Returns

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

Examples

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

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

Merge the body:

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

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

Shell the body to the thickness specified.

Parameters

offset
[Real] Shell thickness.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

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

Shell by removing a given set of faces.

Parameters

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

offset
[Real] Shell thickness.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

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

Plot the body.

Parameters

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

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

use_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.

Module detail

body.**__TEMPORARY_BOOL_OPS_FIX__ = (99, 0, 0)**

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

set_name	Set the name of the component.
get_all_bodies	Get all bodies in the component hierarchy.
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_body_from_surface	Create a surface body from a trimmed surface.
create_surface_from_trimmed_curves	Create a surface body from a list of trimmed curves all lying on the same plane.
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

id	ID of the component.
name	Name of the component.
instance_name	Name of the component instance.
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.**id: str**

ID of the component.

property Component.**name: str**

Name of the component.

property Component.**instance_name: str**

Name of the component instance.

property Component.**components: list[*Component*]**

List of Component objects inside of the component.

property Component.**bodies: list[*ansys.geometry.core.designer.body.Body*]**

List of Body objects inside of the component.

property Component.**beams: list[*ansys.geometry.core.designer.beam.Beam*]**

List of Beam objects inside of the component.

property Component.**design_points:**
list[*ansys.geometry.core.designer.designpoint.DesignPoint*]

List of DesignPoint objects inside of the component.

property Component.**coordinate_systems:**
list[*ansys.geometry.core.designer.coordinate_system.CoordinateSystem*]

List of CoordinateSystem objects inside of the component.

property Component.**parent_component: *Component***

Parent of the component.

property Component.**is_alive: bool**

Whether the component is still alive on the server side.

property Component.**shared_topology: *SharedTopologyType* | None**

Shared topology type of the component (if any).

Notes

If no shared topology has been set, None is returned.

Method detail

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

Set the name of the component.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

Component.**get_all_bodies**() → list[*ansys.geometry.core.designer.body.Body*]

Get all bodies in the component hierarchy.

Returns

list[Body]
List of all bodies in the component hierarchy.

Component.**get_world_transform**() → *ansys.geometry.core.math.matrix.Matrix44*

Get the full transformation matrix of the component in world space.

Returns

Matrix44
4x4 transformation matrix of the component in world space.

Component.**modify_placement**(*translation:* ansys.geometry.core.math.vector.Vector3D | *None* = *None*, *rotation_origin:* ansys.geometry.core.math.point.Point3D | *None* = *None*, *rotation_direction:* ansys.geometry.core.math.vector.UnitVector3D | *None* = *None*, *rotation_angle:* *pint.Quantity* | ansys.geometry.core.misc.measurements.Angle | *ansys.geometry.core.typing.Real* = *0*)

Apply a translation and/or rotation to the placement matrix.

Parameters

translation
[Vector3D, default: None] Vector that defines the desired translation to the component.

rotation_origin
[Point3D, default: None] Origin that defines the axis to rotate the component about.

rotation_direction
[UnitVector3D, default: None] Direction of the axis to rotate the component about.

rotation_angle
[Quantity | Angle | Real, default: 0] Angle to rotate the component around the axis.

Notes

To reset a component's placement to an identity matrix, see *reset_placement()* or call *modify_placement()* with no arguments.

Component.**reset_placement**()

Reset a component's placement matrix to an identity matrix.

See *modify_placement()*.

Component.**add_component**(*name: str*, *template:* Component | *None* = *None*, *instance_name: str* = *None*) → *Component*

Add a new component under this component within the design assembly.

Parameters

name
[str] User-defined label for the new component.

template
[Component, default: None] Template to create this component from. This creates an instance component that shares a master with the template component.

Returns

Component
New component with no children in the design assembly.

Component.**set_shared_topology**(*share_type:* SharedTopologyType) → None

Set the shared topology to apply to the component.

Parameters

share_type
[SharedTopologyType] Shared topology type to assign to the component.

Component.**extrude_sketch**(*name: str*, *sketch:* ansys.geometry.core.sketch.sketch.Sketch, *distance: pint.Quantity* | ansys.geometry.core.misc.measurements.Distance | *ansys.geometry.core.typing.Real*, *direction:* ExtrusionDirection | *str* = *ExtrusionDirection.POSITIVE*, *cut: bool* = *False*) → *ansys.geometry.core.designer.body.Body* | None

Create a solid body by extruding the sketch profile a distance.

Parameters

name
[str] User-defined label for the new solid body.

sketch
[Sketch] Two-dimensional sketch source for the extrusion.

distance
[Quantity | Distance | Real] Distance to extrude the solid body.

direction
[ExtrusionDirection | str, default: "+"] Direction for extruding the solid body. The default is to extrude in the positive normal direction of the sketch. Options are "+" and "-" as a string, or the enum values.

cut
[bool, default: False] Whether to cut the extrusion from the existing component. By default, the extrusion is added to the existing component.

Returns

Body
Extruded body from the given sketch.

None
If the cut parameter is True, the function returns None.

Notes

The newly created body is placed under this component within the design assembly.

Component.**sweep_sketch**(*name: str*, *sketch:* ansys.geometry.core.sketch.sketch.Sketch, *path: list[*ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve*]*) → *ansys.geometry.core.designer.body.Body*

Create a body by sweeping a planar profile along a path.

The newly created body is placed under this component within the design assembly.

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.

Warning

This method is only available starting on Ansys release 24R2.

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.

The newly created body is placed under this component within the design assembly.

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.

Warning

This method is only available starting on Ansys release 24R2.

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.

Warning

This method is only available starting on Ansys release 24R2.

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.

Warning

This method is only available starting on Ansys release 24R2.

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.

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.

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.

Warning

This method is only available starting on Ansys release 24R2.

Component.**create_surface**(*name: str*, *sketch:* ansys.geometry.core.sketch.sketch.Sketch) → *ansys.geometry.core.designer.body.Body*

Create a surface body with a sketch profile.

The newly created body is placed under this component within the design assembly.

Parameters

name
[str] User-defined label for the new surface body.

sketch
[Sketch] Two-dimensional sketch source for the surface definition.

Returns

Body
Body (as a planar surface) from the given sketch.

Component.**create_surface_from_face**(*name: str*, *face:* ansys.geometry.core.designer.face.Face) → *ansys.geometry.core.designer.body.Body*

Create a surface body based on a face.

Parameters

name
[str] User-defined label for the new surface body.

face
[Face] Target face to use as the source for the new surface.

Returns

Body
Surface body.

Notes

The source face can be anywhere within the design component hierarchy. Therefore, there is no validation requiring that the face is placed under the target component where the body is to be created.

Component.**create_body_from_surface**(*name: str*, *trimmed_surface: ansys.geometry.core.shapes.surfaces.TrimmedSurface*) → *ansys.geometry.core.designer.body.Body*

Create a surface body from a trimmed surface.

It is possible to create a closed solid body (as opposed to an open surface body) with a Sphere or Torus if they are untrimmed. This can be validated with *body.is_surface*.

Parameters

name
[str] User-defined label for the new surface body.

trimmed_surface
[TrimmedSurface] Geometry for the new surface body.

Returns

Body
Surface body.

Warning

This method is only available starting on Ansys release 25R1.

Component.**create_surface_from_trimmed_curves**(*name: str*, *trimmed_curves: list[*ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve*]*) → *ansys.geometry.core.designer.body.Body*

Create a surface body from a list of trimmed curves all lying on the same plane.

Parameters

name
[str] User-defined label for the new surface body.

trimmed_curves
[list[TrimmedCurve]] Curves to define the plane and body.

Returns

Body
Surface body.

Warning

This method is only available starting on Ansys release 24R2.

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.

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.

Returns

list[Beam]
A list of the created 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**(*tess_options:* ansys.geometry.core.misc.options.TessellationOptions | *None* = *None*, *_recursive_call: bool* = *False*) → pyvista.PolyData | list[pyvista.MultiBlock]

Tessellate the component.

Parameters

tess_options
[TessellationOptions | None, default: None] A set of options to determine the tessellation quality.

_recursive_call: bool, default: False
Internal flag to indicate if this method is being called recursively. Not to be used by the user.

Returns

PolyData, list[MultiBlock]
Tessellated component as a single PolyData object. If the method is called recursively, a list of MultiBlock objects is returned.

Component.**plot**(*merge_component: bool = True*, *merge_bodies: bool = True*, *screenshot: str | None = None*, *use_trame: bool | None = None*, *use_service_colors: bool | None = None*, *allow_picking: bool | None = None*, ***plotting_options: dict | None*) → None | list[Any]

Plot the component.

Parameters

merge_component
[bool, default: True] Whether to merge the component into a single dataset. By default, True. Performance improved. When True, all the faces of the component are effectively merged into a single dataset. If False, the individual bodies are kept separate.

merge_bodies
[bool, default: True] Whether to merge each body into a single dataset. By default, True. Performance improved. When True, all the faces of each individual body are effectively merged into a single dataset. If False, the individual faces are kept separate.

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

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

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

allow_picking
[bool, default: None] Whether to enable picking. The default is None, in which case the picker is not enabled.

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

Returns

None | list[Any]
If allow_picking=True, a list of picked objects is returned. Otherwise, None.

Examples

Create 25 small cylinders in a grid-like pattern on the XY plane and plot them. Make the cylinders look metallic by enabling physically-based rendering with pbr=True.

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> import numpy as np
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 1, 0])
>>> design = modeler.create_design("my-design")
>>> mycomp = design.add_component("my-comp")
>>> n = 5
>>> xx, yy = np.meshgrid(
...     np.linspace(-4, 4, n),
...     np.linspace(-4, 4, n),
... )
>>> for x, y in zip(xx.ravel(), yy.ravel()):
...     sketch = Sketch(plane)
...     sketch.circle(Point2D([x, y]), 0.2 * u.m)
...     mycomp.extrude_sketch(f"body-{x}-{y}", sketch, 1 * u.m)
>>> mycomp
ansys.geometry.core.designer.Component 0x2203cc9ec50
    Name                 : my-comp
    Exists               : True
    Parent component     : my-design
    N Bodies             : 25
    N Components         : 0
    N Coordinate Systems : 0
>>> mycomp.plot(pbr=True, metallic=1.0)
```

Component.**__repr__**() → str

Represent the Component as a string.

Component.**tree_print**(*consider_comps: bool = True*, *consider_bodies: bool = True*, *consider_beams: bool = True*, *depth: int | None = None*, *indent: int = 4*, *sort_keys: bool = False*, *return_list: bool = False*, *skip_loc_header: bool = False*) → None | list[str]

Print the component in tree format.

Parameters

consider_comps
[bool, default: True] Whether to print the nested components.

consider_bodies
[bool, default: True] Whether to print the bodies.

consider_beams
[bool, default: True] Whether to print the beams.

depth
[int | None, default: None] Depth level to print. If None, it prints all levels.

indent
[int, default: 4] Indentation level. Minimum is 2 - if less than 2, it is set to 2 by default.

sort_keys
[bool, default: False] Whether to sort the keys alphabetically.

return_list
[bool, default: False] Whether to return a list of strings or print out the tree structure.

skip_loc_header
[bool, default: False] Whether to skip the location header. Mostly for internal use.

Returns

None | list[str]
Tree-style printed component or list of strings representing the component tree.

SharedTopologyType

class ansys.geometry.core.designer.component.**SharedTopologyType**(**args*, ***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.

Methods

get_multiplier	Get the multiplier for the extrusion direction.

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.

ExtrusionDirection.**get_multiplier**() → int

Get the multiplier for the extrusion direction.

Returns

int
1 if the direction is positive, -1 if negative.

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.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.

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.

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_disco	Export the design to an dsco file.
export_to_stride	Export the design to an stride 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.
get_all_parameters	Get parameters for the design.
set_parameter	Set or update a parameter of the design.
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.
parameters	List of parameters available for the design.
is_active	Whether the design is currently active.
is_closed	Whether the design is closed (i.e. not active).

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.**parameters: list[*ansys.geometry.core.parameters.parameter.Parameter*]**

List of parameters available for the design.

property Design.**is_active: bool**

Whether the design is currently active.

property Design.**is_closed: bool**

Whether the design is closed (i.e. not active).

Method detail

Design.**close**() → None

Close the design.

Design.**add_material**(*material:* ansys.geometry.core.materials.material.Material) → None

Add a material to the design.

Parameters

material
[Material] Material to add.

Design.**save**(*file_location: pathlib.Path | str*) → None

Save a design to disk on the active Geometry server instance.

Parameters

file_location
[Path | str] Location on disk to save the file to.

Design.**download**(*file_location: pathlib.Path | str*, *format:* DesignFileFormat = *DesignFileFormat.SCDOCX*) → None

Export and download the design from the server.

Parameters

file_location
[Path | str] Location on disk to save the file to.

format
[DesignFileFormat, default: *DesignFileFormat.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_disco**(*location: pathlib.Path | str | None = None*) → pathlib.Path

Export the design to an dsco file.

Parameters

location
[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

Path
The path to the saved file.

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

Export the design to an stride file.

Parameters

location
[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

Path
The path to the saved file.

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

Export the design to a Parasolid text file.

Parameters

location
[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

Path
The path to the saved file.

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

Export the design to a Parasolid binary file.

Parameters

location
[Path | str, optional] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

`Path`
The path to the saved file.

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

Export the design to an FMD file.

Parameters

location
[`Path` | `str`, `optional`] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

`Path`
The path to the saved file.

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

Export the design to a STEP file.

Parameters

location
[`Path` | `str`, `optional`] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

`Path`
The path to the saved file.

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

Export the design to an IGES file.

Parameters

location
[`Path` | `str`, `optional`] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

`Path`
The path to the saved file.

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

Export the design to a PMDB file.

Parameters

location
[`Path` | `str`, `optional`] Location on disk to save the file to. If None, the file will be saved in the current working directory.

Returns

`Path`
The path to the saved file.

Design.**create_named_selection**(*name: str*, *bodies: list[*ansys.geometry.core.designer.body.Body*] | None = None*, *faces: list[*ansys.geometry.core.designer.face.Face*] | None = None*, *edges: list[*ansys.geometry.core.designer.edge.Edge*] | None = None*, *beams: list[*ansys.geometry.core.designer.beam.Beam*] | None = None*, *design_points: list[*ansys.geometry.core.designer.designpoint.DesignPoint*] | None = None*, *components: list[*ansys.geometry.core.designer.component.Component*] | None = None*, *vertices: list[*ansys.geometry.core.designer.vertex.Vertex*] | 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.

components
[list[Component], default: None] All components to include in the named selection.

vertices
[list[Vertex], default: None] All vertices to include in the named selection.

Returns

NamedSelection
Newly created named selection that maintains references to all target entities.

Raises

ValueError
If no entities are provided for the named selection. At least one of the optional parameters must be provided.

Design.**delete_named_selection**(*named_selection:* ansys.geometry.core.designer.selection.NamedSelection | *str*) → None

Delete a named selection on the active Geometry server instance.

Parameters

named_selection
[NamedSelection | str] Name of the named selection or instance.

Design.**delete_component**(*component:* ansys.geometry.core.designer.component.Component | *str*) → None

Delete a component (itself or its children).

Parameters

id
[Union[Component, str]] Name of the component or instance to delete.

Raises

ValueError
The design itself cannot be deleted.

Notes

If the component is not this component (or its children), it is not deleted.

Design.**set_shared_topology**(*share_type:* ansys.geometry.core.designer.component.SharedTopologyType) → None

Set the shared topology to apply to the component.

Parameters

share_type
[SharedTopologyType] Shared topology type to assign.

Raises

ValueError
Shared topology does not apply to a design.

Design.**add_beam_circular_profile**(*name: str*, *radius: pint.Quantity* | ansys.geometry.core.misc.measurements.Distance, *center: numpy.ndarray* | *ansys.geometry.core.typing.RealSequence* | ansys.geometry.core.math.point.Point3D = *ZERO_POINT3D*, *direction_x: numpy.ndarray* | *ansys.geometry.core.typing.RealSequence* | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = *UNITVECTOR3D_X*, *direction_y: numpy.ndarray* | *ansys.geometry.core.typing.RealSequence* | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = *UNITVECTOR3D_Y*) → *ansys.geometry.core.designer.beam.BeamCircularProfile*

Add a new beam circular profile under the design for creating beams.

Parameters

name
[str] User-defined label for the new beam circular profile.

radius
[Quantity | Distance] Radius of the beam circular profile.

center
[ndarray | RealSequence | Point3D] Center of the beam circular profile.

direction_x
[ndarray | RealSequence | UnitVector3D | Vector3D] X-plane direction.

direction_y
[ndarray | RealSequence | UnitVector3D | Vector3D] Y-plane direction.

Design.**get_all_parameters**() → list[*ansys.geometry.core.parameters.parameter.Parameter*]

Get parameters for the design.

Returns

list[Parameter]
List of parameters for the design.

Warning

This method is only available starting on Ansys release 25R1.

`Design.set_parameter`(*dimension:* ansys.geometry.core.parameters.parameter.Parameter) → *ansys.geometry.core.parameters.parameter.ParameterUpdateStatus*

Set or update a parameter of the design.

Parameters

dimension
[`Parameter`] Parameter to set.

Returns

`ParameterUpdateStatus`
Status of the update operation.

Warning

This method is only available starting on Ansys release 25R1.

`Design.add_midsurface_thickness`(*thickness:* *pint.Quantity*, *bodies:* *list*[ansys.geometry.core.designer.body.Body]) → None

Add a mid-surface thickness to a list of bodies.

Parameters

thickness
[`Quantity`] Thickness to be assigned.

bodies
[`list`[Body]] All bodies to include in the mid-surface thickness assignment.

Notes

Only surface bodies will be eligible for mid-surface thickness assignment.

`Design.add_midsurface_offset`(*offset_type:* ansys.geometry.core.designer.body.MidSurfaceOffsetType, *bodies:* *list*[ansys.geometry.core.designer.body.Body]) → None

Add a mid-surface offset type to a list of bodies.

Parameters

offset_type
[`MidSurfaceOffsetType`] Surface offset to be assigned.

bodies
[`list`[Body]] All bodies to include in the mid-surface offset assignment.

Notes

Only surface bodies will be eligible for mid-surface offset assignment.

`Design.delete_beam_profile`(*beam_profile:* ansys.geometry.core.designer.beam.BeamProfile | *str*) → None

Remove a beam profile on the active geometry server instance.

Parameters

beam_profile
[BeamProfile | str] A beam profile name or instance that should be deleted.

Design.**insert_file**(*file_location: pathlib.Path | str*, *import_options:* ansys.geometry.core.misc.options.ImportOptions = *ImportOptions()*) → *ansys.geometry.core.designer.component.Component*

Insert a file into the design.

Parameters

file_location
[Path | str] Location on disk where the file is located.

import_options
[ImportOptions] The options to pass into upload file

Returns

Component
The newly inserted component.

Warning

This method is only available starting on Ansys release 24R2.

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
STRIDE
DISCO
INVALID

Special methods

__str__	Represent object in string format.

Import detail

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

Attribute detail

DesignFileFormat.**SCDOCX** = 'SCDOCX'

DesignFileFormat.**PARASOLID_TEXT** = 'PARASOLID_TEXT'

DesignFileFormat.**PARASOLID_BIN** = 'PARASOLID_BIN'

DesignFileFormat.**FMD** = 'FMD'

DesignFileFormat.**STEP** = 'STEP'

DesignFileFormat.**IGES** = 'IGES'

DesignFileFormat.**PMDB** = 'PMDB'

DesignFileFormat.**STRIDE** = 'STRIDE'

DesignFileFormat.**DISCO** = 'DISCO'

DesignFileFormat.**INVALID** = 'INVALID'

Method detail

DesignFileFormat.**__str__**()

Represent object in string format.

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 | *None* = *None*)

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 | None] Parent component to place the new design point under within the design assembly. Its default value is None.

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.
body	Body 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.
vertices	Vertices that define the edge.
start	Start point of the edge.
end	End point of the edge.
bounding_box	Bounding box of the edge.

Import detail

```
from ansys.geometry.core.designer.edge import Edge
```

Property detail

property Edge.**id: str**

ID of the edge.

property Edge.**body:** *ansys.geometry.core.designer.body.Body*

Body of the edge.

property Edge.**is_reversed: bool**

Flag indicating if the edge is reversed.

property Edge.**shape:** *ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve*

Underlying trimmed curve of the edge.

If the edge is reversed, its shape is the `ReversedTrimmedCurve` type, which swaps the start and end points of the curve and handles parameters to allow evaluation as if the curve is not reversed.

> **Warning**
>
> This method is only available starting on Ansys release 24R2.

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.**vertices:** list[*ansys.geometry.core.designer.vertex.Vertex*]

Vertices that define 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.

property Edge.**bounding_box:** *ansys.geometry.core.math.bbox.BoundingBox*

Bounding box of the edge.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

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

```
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

set_color	Set the color of the face.
set_opacity	Set the opacity of the 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.
create_isoparametric_curves	Create isoparametic curves at the given proportional parameter.
setup_offset_relationship	Create an offset relationship between two faces.
tessellate	Tessellate the face and return the geometry as triangles.
plot	Plot the face.

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.
vertices	List of all vertices of the face.
loops	List of all loops of the face.
color	Get the current color of the face.
opacity	Get the opacity of the face.
bounding_box	Get the bounding box for the face.

Import detail

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

Property detail

property Face.**id: str**

Face ID.

property Face.**is_reversed: bool**

Flag indicating if the face is reversed.

property Face.**body:** *ansys.geometry.core.designer.body.Body*

Body that the face belongs to.

property Face.**shape:** *ansys.geometry.core.shapes.surfaces.trimmed_surface.TrimmedSurface*

Underlying trimmed surface of the face.

If the face is reversed, its shape is a `ReversedTrimmedSurface` type, which handles the direction of the normal vector to ensure it is always facing outward.

> **Warning**
>
> This method is only available starting on Ansys release 24R2.

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.**vertices: list[***ansys.geometry.core.designer.vertex.Vertex***]**

List of all vertices of the face.

property Face.loops: list[*FaceLoop*]

List of all loops of the face.

property Face.color: str

Get the current color of the face.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

property Face.opacity: float

Get the opacity of the face.

property Face.bounding_box: *ansys.geometry.core.math.bbox.BoundingBox*

Get the bounding box for the face.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

Method detail

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

Set the color of the face.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

Face.set_opacity(*opacity: float*) → None

Set the opacity of the face.

> **Warning**
>
> This method is only available starting on Ansys release 25R2.

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.**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.**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.

Face.**setup_offset_relationship**(*other_face:* Face, *set_baselines: bool = False*, *process_adjacent_faces: bool = False*) → bool

Create an offset relationship between two faces.

Parameters

other_face
[Face] The face to setup an offset relationship with.

set_baselines
[bool, default: `False`] Automatically set baseline faces.

process_adjacent_faces
[bool, default: `False`] Look for relationships of the same offset on adjacent faces.

Returns

bool
`True` when successful, `False` when failed.

Warning

This method is only available starting on Ansys release 25R2.

`Face.tessellate`(*tess_options:* ansys.geometry.core.misc.options.TessellationOptions | *None* = *None*) → pyvista.PolyData

Tessellate the face and return the geometry as triangles.

Parameters

tess_options
[`TessellationOptions` | `None`, default: `None`] A set of options to determine the tessellation quality.

Returns

`PolyData`
`pyvista.PolyData` object holding the face.

Notes

The tessellation options are ONLY used if the face has not been tessellated before. If the face has been tessellated before, the stored tessellation is returned.

`Face.plot`(*screenshot: str | None = None*, *use_trame: bool | None = None*, *use_service_colors: bool | None = None*, ***plotting_options: dict | None*) → None

Plot the face.

Parameters

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

use_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 face in the service. The default is `None`, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

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

SurfaceType

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

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

Attribute detail

FaceLoopType.**INNER_LOOP = 'INNER'**

FaceLoopType.**OUTER_LOOP = 'OUTER'**

Description

Module for managing a face.

The geometry_commands.py module

Summary

Classes

GeometryCommands	Provides geometry commands for PyAnsys Geometry.

Enums

ExtrudeType	Provides values for extrusion types.
OffsetMode	Provides values for offset modes during extrusions.
FillPatternType	Provides values for types of fill patterns.

GeometryCommands

class ansys.geometry.core.designer.geometry_commands.**GeometryCommands**(*grpc_client:* ansys.geometry.core.connection.client.GrpcCl

Provides geometry commands for PyAnsys Geometry.

Parameters

grpc_client
[GrpcClient] gRPC client to use for the geometry commands.

Overview

Methods

chamfer	Create a chamfer on an edge or adjust the chamfer of a face.
fillet	Create a fillet on an edge or adjust the fillet of a face.
full_fillet	Create a full fillet betweens a collection of faces.
extrude_faces	Extrude a selection of faces.
extrude_faces_up_to	Extrude a selection of faces up to another object.
extrude_edges	Extrude a selection of edges. Provide either a face or a direction and point.
extrude_edges_up_to	Extrude a selection of edges up to another object.
rename_object	Rename an object.
create_linear_pattern	Create a linear pattern. The pattern can be one or two dimensions.
modify_linear_pattern	Modify a linear pattern. Leave an argument at 0 for it to remain unchanged.
create_circular_pattern	Create a circular pattern. The pattern can be one or two dimensions.
modify_circular_pattern	Modify a circular pattern. Leave an argument at 0 for it to remain unchanged.
create_fill_pattern	Create a fill pattern.
update_fill_pattern	Update a fill pattern.
revolve_faces	Revolve face around an axis.
revolve_faces_up_to	Revolve face around an axis up to a certain object.
revolve_faces_by_helix	Revolve face around an axis in a helix shape.
replace_face	Replace a face with another face.
split_body	Split bodies with a plane, slicers, or faces.
get_round_info	Get info on the rounding of a face.
move_translate	Move a selection by a distance in a direction.
move_rotate	Rotate a selection by an angle about a given axis.
offset_faces_set_radius	Offset faces with a radius.
create_align_condition	Create an align condition between two geometry objects.
create_tangent_condition	Create a tangent condition between two geometry objects.
create_orient_condition	Create an orient condition between two geometry objects.

Import detail

```
from ansys.geometry.core.designer.geometry_commands import GeometryCommands
```

Method detail

GeometryCommands.**chamfer**(*selection:* ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge] | ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *distance:* *ansys.geometry.core.typing.Real*) → bool

Create a chamfer on an edge or adjust the chamfer of a face.

Parameters

selection
[Edge | list[Edge] | Face | list[Face]] One or more edges or faces to act on.

distance
[Real] Chamfer distance.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**fillet**(*selection:* ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge] | ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *radius: ansys.geometry.core.typing.Real*) → bool

Create a fillet on an edge or adjust the fillet of a face.

Parameters

selection
[Edge | list[Edge] | Face | list[Face]] One or more edges or faces to act on.

radius
[Real] Fillet radius.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**full_fillet**(*faces: list*[ansys.geometry.core.designer.face.Face]) → bool

Create a full fillet betweens a collection of faces.

Parameters

faces
[list[Face]] Faces to round.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**extrude_faces**(*faces:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *distance: ansys.geometry.core.typing.Real*, *direction:* ansys.geometry.core.math.vector.UnitVector3D = *None*, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*, *offset_mode:* OffsetMode = *OffsetMode.MOVE_FACES_TOGETHER*, *pull_symmetric: bool* = *False*, *copy: bool* = *False*, *force_do_as_extrude: bool* = *False*) → list[*ansys.geometry.core.designer.body.Body*]

Extrude a selection of faces.

Parameters

faces
[Face | list[Face]] Faces to extrude.

distance
[Real] Distance to extrude.

direction
[UnitVector3D, default: None] Direction of extrusion. If no direction is provided, it will be inferred.

extrude_type
[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

offset_mode
[OffsetMode, default: *OffsetMode.MOVE_FACES_TOGETHER*] Mode of how to handle offset relationships.

pull_symmetric
[bool, default: False] Pull symmetrically on both sides if True.

copy
[bool, default: False] Copy the face and move it instead of extruding the original face if True.

force_do_as_extrude
[bool, default: False] Forces to do as an extrusion if True, if False allows extrusion by offset.

Returns

list[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**extrude_faces_up_to**(*faces:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *up_to_selection:* ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.body.Body, *seed_point:* ansys.geometry.core.math.point.Point3D, *direction:* ansys.geometry.core.math.vector.UnitVector3D, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*, *offset_mode:* OffsetMode = *OffsetMode.MOVE_FACES_TOGETHER*, *pull_symmetric:* *bool* = *False*, *copy:* *bool* = *False*, *force_do_as_extrude:* *bool* = *False*) → list[*ansys.geometry.core.designer.body.Body*]

Extrude a selection of faces up to another object.

Parameters

faces
[Face | list[Face]] Faces to extrude.

up_to_selection
[Face | Edge | Body] The object to pull the faces up to.

seed_point
[Point3D] Origin to define the extrusion.

direction
[UnitVector3D, default: None] Direction of extrusion. If no direction is provided, it will be inferred.

extrude_type
[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

offset_mode
[OffsetMode, default: *OffsetMode.MOVE_FACES_TOGETHER*] Mode of how to handle offset relationships.

pull_symmetric
[bool, default: False] Pull symmetrically on both sides if True.

copy
[bool, default: False] Copy the face and move it instead of extruding the original face if True.

force_do_as_extrude
[bool, default: False] Forces to do as an extrusion if True, if False allows extrusion by offset.

Returns

list[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**extrude_edges**(*edges:* ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge], *distance:* *ansys.geometry.core.typing.Real*, *from_face:* ansys.geometry.core.designer.face.Face = *None*, *from_point:* ansys.geometry.core.math.point.Point3D = *None*, *direction:* ansys.geometry.core.math.vector.UnitVector3D = *None*, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*, *pull_symmetric:* *bool* = *False*, *copy:* *bool* = *False*, *natural_extension:* *bool* = *False*) → list[*ansys.geometry.core.designer.body.Body*]

Extrude a selection of edges. Provide either a face or a direction and point.

Parameters

edges
[Edge | list[Edge]] Edges to extrude.

distance
[Real] Distance to extrude.

from_face
[Face, default: None] Face to pull normal from.

from_point
[Point3D, default: None] Point to pull from. Must be used with direction.

direction
[UnitVector3D, default: None] Direction to pull. Must be used with from_point.

extrude_type
[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

pull_symmetric
[bool, default: False] Pull symmetrically on both sides if True.

copy
[bool, default: False] Copy the edge and move it instead of extruding the original edge if True.

natural_extension
[bool, default: False] Surfaces will extend in a natural or linear shape after exceeding its original range.

Returns

list[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**extrude_edges_up_to**(*edges:* ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge], *up_to_selection:* ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.body.Body, *seed_point:* ansys.geometry.core.math.point.Point3D, *direction:* ansys.geometry.core.math.vector.UnitVector3D, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*) → list[*ansys.geometry.core.designer.body.Body*]

Extrude a selection of edges up to another object.

Parameters

edges
[Edge | list[Edge]] Edges to extrude.

up_to_selection
[Face, default: None] The object to pull the faces up to.

seed_point
[Point3D] Origin to define the extrusion.

direction
[UnitVector3D, default: None] Direction of extrusion.

extrude_type
[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

Returns

list[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**rename_object**(*selection: list[ansys.geometry.core.designer.body.Body] | list[ansys.geometry.core.designer.component.Component] | list[ansys.geometry.core.designer.face.Face] | list[ansys.geometry.core.designer.edge.Edge]*, *name: str*) → bool

Rename an object.

Parameters

selection
[list[Body] | list[Component] | list[Face] | list[Edge]] Selection of the object to rename.

name
[str] New name for the object.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**create_linear_pattern**(*selection: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face]*, *linear_direction: ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.face.Face*, *count_x: int*, *pitch_x: ansys.geometry.core.typing.Real*, *two_dimensional: bool = False*, *count_y: int = None*, *pitch_y: ansys.geometry.core.typing.Real = None*) → bool

Create a linear pattern. The pattern can be one or two dimensions.

Parameters

selection
[Face | list[Face]] Faces to create the pattern out of.

linear_direction
[Edge | Face] Direction of the linear pattern, determined by the direction of an edge or face normal.

count_x
[int] How many times the pattern repeats in the x direction.

pitch_x
[Real] The spacing between each pattern member in the x direction.

two_dimensional
[bool, default: False] If True, create a pattern in the x and y direction.

count_y
[int, default: None] How many times the pattern repeats in the y direction.

pitch_y
[Real, default: None] The spacing between each pattern member in the y direction.

Returns

bool
True when successful, False when failed.

GeometryCommands.**modify_linear_pattern**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *count_x:* *int* = *0*, *pitch_x:* *ansys.geometry.core.typing.Real* = *0.0*, *count_y:* *int* = *0*, *pitch_y:* *ansys.geometry.core.typing.Real* = *0.0*, *new_seed_index:* *int* = *0*, *old_seed_index:* *int* = *0*) → bool

Modify a linear pattern. Leave an argument at 0 for it to remain unchanged.

Parameters

selection
[Face | list[Face]] Faces that belong to the pattern.

count_x
[int, default: 0] How many times the pattern repeats in the x direction.

pitch_x
[Real, default: 0.0] The spacing between each pattern member in the x direction.

count_y
[int, default: 0] How many times the pattern repeats in the y direction.

pitch_y
[Real, default: 0.0] The spacing between each pattern member in the y direction.

new_seed_index
[int, default: 0] The new seed index of the member.

old_seed_index
[int, default: 0] The old seed index of the member.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**create_circular_pattern**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *circular_axis:* ansys.geometry.core.designer.edge.Edge, *circular_count:* *int*, *circular_angle:* *ansys.geometry.core.typing.Real*, *two_dimensional:* *bool* = *False*, *linear_count:* *int* = *None*, *linear_pitch:* *ansys.geometry.core.typing.Real* = *None*, *radial_direction:* ansys.geometry.core.math.vector.UnitVector3D = *None*) → bool

Create a circular pattern. The pattern can be one or two dimensions.

Parameters

selection
[Face | list[Face]] Faces to create the pattern out of.

circular_axis
[Edge] The axis of the circular pattern, determined by the direction of an edge.

circular_count
[int] How many members are in the circular pattern.

circular_angle
[Real] The angular range of the pattern.

two_dimensional
[bool, default: False] If True, create a two-dimensional pattern.

linear_count
[int, default: None] How many times the circular pattern repeats along the radial lines for a two-dimensional pattern.

linear_pitch
[Real, default: None] The spacing along the radial lines for a two-dimensional pattern.

radial_direction
[UnitVector3D, default: None] The direction from the center out for a two-dimensional pattern.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**modify_circular_pattern**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *circular_count:* *int* = *0*, *linear_count:* *int* = *0*, *step_angle:* *ansys.geometry.core.typing.Real* = *0.0*, *step_linear:* *ansys.geometry.core.typing.Real* = *0.0*) → bool

Modify a circular pattern. Leave an argument at 0 for it to remain unchanged.

Parameters

selection
[Face | list[Face]] Faces that belong to the pattern.

circular_count
[int, default: 0] How many members are in the circular pattern.

linear_count
[int, default: 0] How many times the circular pattern repeats along the radial lines for a two-dimensional pattern.

step_angle
[Real, default: 0.0] Defines the circular angle.

step_linear
[Real, default: 0.0] Defines the step, along the radial lines, for a pattern dimension greater than 1.

Returns

bool
`True` when successful, `False` when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**create_fill_pattern**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *linear_direction:* ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.face.Face, *fill_pattern_type:* FillPatternType, *margin: ansys.geometry.core.typing.Real*, *x_spacing: ansys.geometry.core.typing.Real*, *y_spacing: ansys.geometry.core.typing.Real*, *row_x_offset: ansys.geometry.core.typing.Real = 0*, *row_y_offset: ansys.geometry.core.typing.Real = 0*, *column_x_offset: ansys.geometry.core.typing.Real = 0*, *column_y_offset: ansys.geometry.core.typing.Real = 0*) → bool

Create a fill pattern.

Parameters

selection
[`Face` | `list`[`Face`]] Faces to create the pattern out of.

linear_direction
[`Edge`] Direction of the linear pattern, determined by the direction of an edge.

fill_pattern_type
[`FillPatternType`] The type of fill pattern.

margin
[`Real`] Margin defining the border of the fill pattern.

x_spacing
[`Real`] Spacing between the pattern members in the x direction.

y_spacing
[`Real`] Spacing between the pattern members in the x direction.

row_x_offset
[`Real`, default: 0] Offset for the rows in the x direction. Only used with `FillPattern.SKEWED`.

row_y_offset
[`Real`, default: 0] Offset for the rows in the y direction. Only used with `FillPattern.SKEWED`.

column_x_offset
[`Real`, default: 0] Offset for the columns in the x direction. Only used with `FillPattern.SKEWED`.

column_y_offset
[`Real`, default: 0] Offset for the columns in the y direction. Only used with `FillPattern.SKEWED`.

Returns

bool
`True` when successful, `False` when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**update_fill_pattern**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face]) → bool

Update a fill pattern.

When the face that a fill pattern exists upon changes in size, the fill pattern can be updated to fill the new space.

Parameters

selection
[`Face` | `list`[`Face`]] Face(s) that are part of a fill pattern.

Returns

bool
`True` when successful, `False` when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**revolve_faces**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *axis:* ansys.geometry.core.shapes.curves.line.Line, *angle:* *ansys.geometry.core.typing.Real*, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*) → list[*ansys.geometry.core.designer.body.Body*]

Revolve face around an axis.

Parameters

selection
[`Face` | `list`[`Face`]] Face(s) to revolve.

axis
[`Line`] Axis of revolution.

angle
[`Real`] Angular distance to revolve.

extrude_type
[`ExtrudeType`, default: *`ExtrudeType.ADD`*] Type of extrusion to be performed.

Returns

`list`[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**revolve_faces_up_to**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *up_to:* ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.designer.body.Body, *axis:* ansys.geometry.core.shapes.curves.line.Line, *direction:* ansys.geometry.core.math.vector.UnitVector3D, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*) → list[*ansys.geometry.core.designer.body.Body*]

Revolve face around an axis up to a certain object.

Parameters

selection
[Face | list[Face]] Face(s) to revolve.

up_to
[Face | Edge | Body] Object to revolve the face up to.

axis
[Line] Axis of revolution.

direction
[UnitVector3D] Direction of extrusion.

extrude_type
[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

Returns

list[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**revolve_faces_by_helix**(*selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *axis:* ansys.geometry.core.shapes.curves.line.Line, *direction:* ansys.geometry.core.math.vector.UnitVector3D, *height:* *ansys.geometry.core.typing.Real*, *pitch:* *ansys.geometry.core.typing.Real*, *taper_angle:* *ansys.geometry.core.typing.Real*, *right_handed:* *bool*, *both_sides:* *bool*, *extrude_type:* ExtrudeType = *ExtrudeType.ADD*) → list[*ansys.geometry.core.designer.body.Body*]

Revolve face around an axis in a helix shape.

Parameters

selection
[Face | list[Face]] Face(s) to revolve.

axis
[Line] Axis of revolution.

direction
[UnitVector3D] Direction of extrusion.

height
[Real,] Height of the helix.

pitch
[Real,] Pitch of the helix.

taper_angle
[Real,] Tape angle of the helix.

right_handed
[bool,] Right-handed helix if `True`, left-handed if `False`.

both_sides
[bool,] Create on both sides if `True`, one side if `False`.

extrude_type
[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

Returns

list[Body]
Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**replace_face**(*target_selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *replacement_selection:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face]) → bool

Replace a face with another face.

Parameters

target_selection
[Union[Face, list[Face]]] The face or faces to replace.

replacement_selection
[Union[Face, list[Face]]] The face or faces to replace with.

Returns

bool
`True` when successful, `False` when failed.

GeometryCommands.**split_body**(*bodies:* *list*[ansys.geometry.core.designer.body.Body], *plane:* ansys.geometry.core.math.plane.Plane, *slicers:* ansys.geometry.core.designer.edge.Edge | *list*[ansys.geometry.core.designer.edge.Edge] | ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *faces:* *list*[ansys.geometry.core.designer.face.Face], *extendfaces:* *bool*) → bool

Split bodies with a plane, slicers, or faces.

Parameters

bodies
[list[Body]] Bodies to split.

plane
[Plane] Plane to split with

slicers
[Edge | list[Edge] | Face | list[Face]] Slicers to split with.

faces
[list[Face]] Faces to split with.

extendFaces
[bool] Extend faces if split with faces.

Returns

bool
`True` when successful, `False` when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**get_round_info**(*face:* ansys.geometry.core.designer.face.Face) → tuple[bool, ansys.geometry.core.typing.Real]

Get info on the rounding of a face.

Parameters

Face
The design face to get round info on.

Returns

tuple[bool, Real]
`True` if round is aligned with face's U-parameter direction, `False` otherwise. Radius of the round.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**move_translate**(*selection:* ansys.geometry.core.designer.selection.NamedSelection, *direction:* ansys.geometry.core.math.vector.UnitVector3D, *distance:* ansys.geometry.core.misc.measurements.Distance | *pint.Quantity* | *ansys.geometry.core.typing.Real*) → bool

Move a selection by a distance in a direction.

Parameters

selection
[NamedSelection] Named selection to move.

direction
[UnitVector3D] Direction to move in.

distance
[Distance | Quantity | Real] Distance to move. Default units are meters.

Returns

bool
True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**move_rotate**(*selection:* ansys.geometry.core.designer.selection.NamedSelection, *axis:* ansys.geometry.core.shapes.curves.line.Line, *angle:* ansys.geometry.core.misc.measurements.Angle | *pint.Quantity* | *ansys.geometry.core.typing.Real*) → dict[str, bool | ansys.geometry.core.typing.Real]

Rotate a selection by an angle about a given axis.

Parameters

selection
[NamedSelection] Named selection to move.

axis
[Line] Direction to move in.

Angle
[Angle | Quantity | Real] Angle to rotate by. Default units are radians.

Returns

dict[str, Union[bool, Real]]
Dictionary containing the useful output from the command result. Keys are success, modified_bodies, modified_faces, modified_edges.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**offset_faces_set_radius**(*faces:* ansys.geometry.core.designer.face.Face | *list*[ansys.geometry.core.designer.face.Face], *radius:* ansys.geometry.core.misc.measurements.Distance | *pint.Quantity* | *ansys.geometry.core.typing.Real*, *copy:* *bool* = *False*, *offset_mode:* OffsetMode = *OffsetMode.IGNORE_RELATIONSHIPS*, *extrude_type:* ExtrudeType = *ExtrudeType.FORCE_INDEPENDENT*) → bool

Offset faces with a radius.

Parameters

faces
[Face | list[Face]] Faces to offset.

radius
[Distance | Quantity | Real] Radius of the offset.

copy
[bool, default: `False`] Copy the face and move it instead of offsetting the original face if `True`.

offset_mode
[`OffsetMode`, default: *`OffsetMode.MOVE_FACES_TOGETHER`*] Mode of how to handle offset relationships.

extrude_type
[`ExtrudeType`, default: *`ExtrudeType.FORCE_INDEPENDENT`*] Type of extrusion to be performed.

Returns

bool
`True` when successful, `False` when failed.

Warning

This method is only available starting on Ansys release 25R2.

`GeometryCommands.`**`create_align_condition`**(*parent_component:* ansys.geometry.core.designer.component.Component, *geometry_a:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge, *geometry_b:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge) → *ansys.geometry.core.designer.mating_conditions.AlignCondition*

Create an align condition between two geometry objects.

This will move the objects to be aligned with each other.

Parameters

parent_component
[`Component`] The common ancestor component of the two geometry objects.

geometry_a
[`Body` | `Face` | `Edge`] The first geometry object to align to the second.

geometry_b
[`Body` | `Face` | `Edge`] The geometry object to be aligned to.

Returns

`AlignCondition`
The persistent align condition that was created.

Warning

This method is only available starting on Ansys release 26R1.

GeometryCommands.**create_tangent_condition**(*parent_component:* ansys.geometry.core.designer.component.Component, *geometry_a:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge, *geometry_b:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge) → *ansys.geometry.core.designer.mating_conditions.TangentCondition*

Create a tangent condition between two geometry objects.

This aligns the objects so that they are tangent.

Parameters

parent_component
[Component] The common ancestor component of the two geometry objects.

geometry_a
[Body | Face | Edge] The first geometry object to tangent the second.

geometry_b
[Body | Face | Edge] The geometry object to be tangent with.

Returns

TangentCondition
The persistent tangent condition that was created.

Warning

This method is only available starting on Ansys release 26R1.

GeometryCommands.**create_orient_condition**(*parent_component:* ansys.geometry.core.designer.component.Component, *geometry_a:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge, *geometry_b:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge) → *ansys.geometry.core.designer.mating_conditions.OrientCondition*

Create an orient condition between two geometry objects.

This rotates the objects so that they are oriented in the same direction.

Parameters

parent_component
[Component] The common ancestor component of the two geometry objects.

geometry_a
[Body | Face | Edge] The first geometry object to orient with the second.

geometry_b
[Body | Face | Edge] The geometry object to be oriented with.

Returns

OrientCondition
The persistent orient condition that was created.

> **Warning**
>
> This method is only available starting on Ansys release 26R1.

ExtrudeType

class ansys.geometry.core.designer.geometry_commands.**ExtrudeType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for extrusion types.

Overview

Attributes

NONE
ADD
CUT
FORCE_ADD
FORCE_CUT
FORCE_INDEPENDENT
FORCE_NEW_SURFACE

Import detail

```
from ansys.geometry.core.designer.geometry_commands import ExtrudeType
```

Attribute detail

ExtrudeType.**NONE = 0**

ExtrudeType.**ADD = 1**

ExtrudeType.**CUT = 2**

ExtrudeType.**FORCE_ADD = 3**

ExtrudeType.**FORCE_CUT = 4**

ExtrudeType.**FORCE_INDEPENDENT = 5**

ExtrudeType.**FORCE_NEW_SURFACE = 6**

OffsetMode

class ansys.geometry.core.designer.geometry_commands.**OffsetMode**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for offset modes during extrusions.

Overview

Attributes

IGNORE_RELATIONSHIPS
MOVE_FACES_TOGETHER
MOVE_FACES_APART

Import detail

```
from ansys.geometry.core.designer.geometry_commands import OffsetMode
```

Attribute detail

OffsetMode.**IGNORE_RELATIONSHIPS = 0**

OffsetMode.**MOVE_FACES_TOGETHER = 1**

OffsetMode.**MOVE_FACES_APART = 2**

FillPatternType

class ansys.geometry.core.designer.geometry_commands.**FillPatternType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for types of fill patterns.

Overview

Attributes

GRID
OFFSET
SKEWED

Import detail

```
from ansys.geometry.core.designer.geometry_commands import FillPatternType
```

Attribute detail

FillPatternType.**GRID = 0**

FillPatternType.**OFFSET = 1**

FillPatternType.**SKEWED = 2**

Description

Provides tools for pulling geometry.

The `mating_conditions.py` module

Summary

Classes

MatingCondition	MatingCondition dataclass.
AlignCondition	AlignCondition dataclass.
TangentCondition	TangentCondition dataclass.
OrientCondition	OrientCondition dataclass.

MatingCondition

class ansys.geometry.core.designer.mating_conditions.**MatingCondition**

MatingCondition dataclass.

Overview

Attributes

id
is_deleted
is_enabled
is_satisfied

Import detail

```
from ansys.geometry.core.designer.mating_conditions import MatingCondition
```

Attribute detail

MatingCondition.**id: str**

MatingCondition.**is_deleted: bool**

MatingCondition.**is_enabled: bool**

MatingCondition.**is_satisfied: bool**

AlignCondition

class ansys.geometry.core.designer.mating_conditions.**AlignCondition**

Bases: MatingCondition

AlignCondition dataclass.

Overview

Attributes

offset
is_reversed
is_valid

Import detail

```
from ansys.geometry.core.designer.mating_conditions import AlignCondition
```

Attribute detail

AlignCondition.**offset: float**

AlignCondition.**is_reversed: bool**

AlignCondition.**is_valid: bool**

TangentCondition

class ansys.geometry.core.designer.mating_conditions.**TangentCondition**

Bases: MatingCondition

TangentCondition dataclass.

Overview

Attributes

offset
is_reversed
is_valid

Import detail

```
from ansys.geometry.core.designer.mating_conditions import TangentCondition
```

Attribute detail

TangentCondition.**offset: float**

TangentCondition.**is_reversed: bool**

TangentCondition.**is_valid: bool**

OrientCondition

class ansys.geometry.core.designer.mating_conditions.**OrientCondition**

Bases: MatingCondition

OrientCondition dataclass.

Overview

Attributes

offset
is_reversed
is_valid

Import detail

```
from ansys.geometry.core.designer.mating_conditions import OrientCondition
```

Attribute detail

OrientCondition.**offset: float**

OrientCondition.**is_reversed: bool**

OrientCondition.**is_valid: bool**

Description

Provides dataclasses for mating conditions.

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

```
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*, *design:* ansys.geometry.core.designer.design.Design, *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*, *components:* *list[*ansys.geometry.core.designer.component.Componen | *None* = *None*, *vertices:* *list[*ansys.geometry.core.designer.vertex.Vertex*]* | *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.

design
[Design] Design instance to which the named selection belongs.

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.

components: list[Component], default: None
All components to include in the named selection.

Overview

Properties

id	ID of the named selection.
name	Name of the named selection.
bodies	All bodies in the named selection.
faces	All faces in the named selection.
edges	All edges in the named selection.
beams	All beams in the named selection.
design_points	All design points in the named selection.
components	All components in the named selection.
vertices	All vertices in the named selection.

Special methods

__repr__	Represent the NamedSelection as a string.

Import detail

```
from ansys.geometry.core.designer.selection import NamedSelection
```

Property detail

property NamedSelection.**id: str**

ID of the named selection.

property NamedSelection.**name: str**

Name of the named selection.

property NamedSelection.**bodies**: list[*ansys.geometry.core.designer.body.Body*]

All bodies in the named selection.

property NamedSelection.**faces**: list[*ansys.geometry.core.designer.face.Face*]

All faces in the named selection.

property NamedSelection.**edges**: list[*ansys.geometry.core.designer.edge.Edge*]

All edges in the named selection.

property NamedSelection.**beams**: list[*ansys.geometry.core.designer.beam.Beam*]

All beams in the named selection.

property NamedSelection.**design_points**: list[*ansys.geometry.core.designer.designpoint.DesignPoint*]

All design points in the named selection.

property NamedSelection.**components**: list[*ansys.geometry.core.designer.component.Component*]

All components in the named selection.

property NamedSelection.**vertices**: list[*ansys.geometry.core.designer.vertex.Vertex*]

All vertices in the named selection.

Method detail

NamedSelection.**__repr__**() → str

Represent the NamedSelection as a string.

Description

Module for creating a named selection.

The vertex.py module

Summary

Classes

Vertex	Represents a single vertex of a body within the design assembly.

Vertex

class ansys.geometry.core.designer.vertex.**Vertex**(*id: str*, *position: numpy.ndarray | ansys.geometry.core.typing.RealSequence*)

Bases: *ansys.geometry.core.math.point.Point3D*

Represents a single vertex of a body within the design assembly.

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

Parameters

id
[str] The unique identifier for the vertex.

position
[np.ndarray or RealSequence] The position of the vertex in 3D space.

Overview

Properties

id	Get the unique identifier of the vertex.

Special methods

__repr__	Return a string representation of the vertex.

Import detail

```
from ansys.geometry.core.designer.vertex import Vertex
```

Property detail

property Vertex.**id**: str

Get the unique identifier of the vertex.

Method detail

Vertex.**__repr__**() → str

Return a string representation of the vertex.

Description

Module for managing a vertex.

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.
BoundingBox	Maintains the box structure for Bounding Boxes.

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.

Static methods

intersect_bboxes	Find the intersection of 2 BoundingBox2D objects.

Special methods

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

Import detail

```
from ansys.geometry.core.math.bbox import BoundingBox2D
```

Property detail

property BoundingBox2D.**x_min: ansys.geometry.core.typing.Real**

Minimum value of X-dimensional bounds.

Returns

Real
Minimum value of the X-dimensional bounds.

property BoundingBox2D.**x_max: ansys.geometry.core.typing.Real**

Maximum value of the X-dimensional bounds.

Returns

Real
Maximum value of the X-dimensional bounds.

property BoundingBox2D.**y_min: ansys.geometry.core.typing.Real**

Minimum value of Y-dimensional bounds.

Returns

Real
Minimum value of Y-dimensional bounds.

property BoundingBox2D.**y_max: ansys.geometry.core.typing.Real**

Maximum value of Y-dimensional bounds.

Returns

Real
Maximum value of Y-dimensional bounds.

Method detail

BoundingBox2D.**add_point**(*point:* ansys.geometry.core.math.point.Point2D) → None

Extend the ranges of the bounding box to include a point.

Parameters

point
[Point2D] Point to include within the bounds.

Notes

This method is only applicable if the point components are outside the current bounds.

BoundingBox2D.**add_point_components**(*x: ansys.geometry.core.typing.Real*, *y: ansys.geometry.core.typing.Real*) → None

Extend the ranges of the bounding box to include the X and Y values.

Parameters

x
[Real] Point X component to include within the bounds.

y
[Real] Point Y component to include within the bounds.

Notes

This method is only applicable if the point components are outside the current bounds.

BoundingBox2D.**add_points**(*points: list[*ansys.geometry.core.math.point.Point2D*]*) → None

Extend the ranges of the bounding box to include given points.

Parameters

points
[list[Point2D]] List of points to include within the bounds.

BoundingBox2D.**contains_point**(*point:* ansys.geometry.core.math.point.Point2D) → bool

Evaluate whether a point lies within the X and Y range bounds.

Parameters

point
[Point2D] Point to compare against the bounds.

Returns

bool
True if the point is contained in the bounding box. Otherwise, False.

BoundingBox2D.**contains_point_components**(*x: ansys.geometry.core.typing.Real*, *y: ansys.geometry.core.typing.Real*) → bool

Check if point components are within the X and Y range bounds.

Parameters

x
[Real] Point X component to compare against the bounds.

y
[Real] Point Y component to compare against the bounds.

Returns

bool
True if the components are contained in the bounding box. Otherwise, False.

BoundingBox2D.**__eq__**(*other:* BoundingBox2D) → bool

Equals operator for the BoundingBox2D class.

BoundingBox2D.**__ne__**(*other:* BoundingBox2D) → bool

Not equals operator for the BoundingBox2D class.

static BoundingBox2D.**intersect_bboxes**(*box_1:* BoundingBox2D, *box_2:* BoundingBox2D) → None | *BoundingBox2D*

Find the intersection of 2 BoundingBox2D objects.

Parameters

box_1: BoundingBox2D
The box to consider the intersection of with respect to box_2.

box_2: BoundingBox2D
The box to consider the intersection of with respect to box_1.

Returns

BoundingBox2D:
The box representing the intersection of the two passed in boxes.

BoundingBox

class ansys.geometry.core.math.bbox.**BoundingBox**(*min_corner:* ansys.geometry.core.math.point.Point3D, *max_corner:* ansys.geometry.core.math.point.Point3D, *center:* ansys.geometry.core.math.point.Point3D = *None*)

Maintains the box structure for Bounding Boxes.

Parameters

min_corner
[Point3D] Minimum corner for the box.

max_corner
[Point3D] Maximum corner for the box.

center
[Point3D] Center of the box.

Overview

Methods

contains_point	Evaluate whether a point lies within the box.
contains_point_components	Check if point components are within box.

Properties

min_corner	Minimum corner of the bounding box.
max_corner	Maximum corner of the bounding box.
center	Center of the bounding box.

Static methods

intersect_bboxes	Find the intersection of 2 BoundingBox objects.

Special methods

__eq__	Equals operator for the BoundingBox class.
__ne__	Not equals operator for the BoundingBox class.

Import detail

```
from ansys.geometry.core.math.bbox import BoundingBox
```

Property detail

property BoundingBox.**min_corner:** *ansys.geometry.core.math.point.Point3D*

Minimum corner of the bounding box.

property BoundingBox.**max_corner:** *ansys.geometry.core.math.point.Point3D*

Maximum corner of the bounding box.

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

Center of the bounding box.

Method detail

BoundingBox.**contains_point**(*point:* ansys.geometry.core.math.point.Point3D) → bool

Evaluate whether a point lies within the box.

Parameters

point
[Point3D] Point to compare against the bounds.

Returns

bool
True if the point is contained in the bounding box. Otherwise, False.

BoundingBox.**contains_point_components**(*x: ansys.geometry.core.typing.Real*, *y: ansys.geometry.core.typing.Real*, *z: ansys.geometry.core.typing.Real*) → bool

Check if point components are within box.

Parameters

x
[Real] Point X component to compare against the bounds.

y
[Real] Point Y component to compare against the bounds.

z
[Real] Point Z component to compare against the bounds.

Returns

bool
True if the components are contained in the bounding box. Otherwise, False.

BoundingBox.**__eq__**(*other:* BoundingBox) → bool

Equals operator for the BoundingBox class.

BoundingBox.**__ne__**(*other:* BoundingBox) → bool

Not equals operator for the BoundingBox class.

static BoundingBox.**intersect_bboxes**(*box_1:* BoundingBox, *box_2:* BoundingBox) → None | *BoundingBox*

Find the intersection of 2 BoundingBox objects.

Parameters

box_1: BoundingBox
The box to consider the intersection of with respect to box_2.

box_2: BoundingBox
The box to consider the intersection of with respect to box_1.

Returns

BoundingBox:
The box representing the intersection of the two passed in boxes.

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.

Overview

Constructors

create_translation	Create a matrix representing the specified translation.
create_rotation	Create a matrix representing the specified rotation.
create_matrix_from_rotation_about_axis	Create a matrix representing a rotation about a given axis.
create_matrix_from_mapping	Create a matrix representing the specified mapping.

Methods

is_translation	Check if the matrix represents a translation.

Import detail

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

Method detail

classmethod Matrix44.**create_translation**(*translation:* ansys.geometry.core.math.vector.Vector3D) → *Matrix44*

Create a matrix representing the specified translation.

Parameters

translation
[Vector3D] The translation vector representing the translation. The components of the vector should be in meters.

Returns

Matrix44
A 4x4 matrix representing the translation.

Examples

```
>>> translation_vector = Vector3D(1.0, 2.0, 3.0)
>>> translation_matrix = Matrix44.create_translation(translation_vector)
>>> print(translation_matrix)
[[1. 0. 0. 1.]
 [0. 1. 0. 2.]
 [0. 0. 1. 3.]
 [0. 0. 0. 1.]]
```

Matrix44.**is_translation**(*including_identity: bool = False*) → bool

Check if the matrix represents a translation.

This method checks if the matrix represents a translation transformation. A translation matrix has the following form:

[1 0 0 tx] [0 1 0 ty] [0 0 1 tz] [0 0 0 1]

Parameters

including_identity
[bool, optional] If True, the method will return True for the identity matrix as well. If False, the method will return False for the identity matrix.

Returns

bool
True if the matrix represents a translation, False otherwise.

Examples

```
>>> matrix = Matrix44([[1, 0, 0, 5], [0, 1, 0, 3], [0, 0, 1, 2], [0, 0, 0, 1]])
>>> matrix.is_translation()
True
>>> identity_matrix = Matrix44([[1, 0, 0, 0], [0, 1, 0, 0], [0, 0, 1, 0], [0, 0, 0,␣
```

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```
↪1]])
>>> identity_matrix.is_translation()
False
>>> identity_matrix.is_translation(including_identity=True)
True
```

classmethod Matrix44.**create_rotation**(*direction_x:* ansys.geometry.core.math.vector.Vector3D, *direction_y:* ansys.geometry.core.math.vector.Vector3D, *direction_z:* ansys.geometry.core.math.vector.Vector3D = *None*) → *Matrix44*

Create a matrix representing the specified rotation.

Parameters

direction_x
[Vector3D] The X direction vector.

direction_y
[Vector3D] The Y direction vector.

direction_z
[Vector3D, optional] The Z direction vector. If not provided, it will be calculated as the cross product of direction_x and direction_y.

Returns

Matrix44
A 4x4 matrix representing the rotation.

Examples

```
>>> direction_x = Vector3D(1.0, 0.0, 0.0)
>>> direction_y = Vector3D(0.0, 1.0, 0.0)
>>> rotation_matrix = Matrix44.create_rotation(direction_x, direction_y)
>>> print(rotation_matrix)
[[1. 0. 0. 0.]
[0. 1. 0. 0.]
[0. 0. 1. 0.]
[0. 0. 0. 1.]]
```

classmethod Matrix44.**create_matrix_from_rotation_about_axis**(*axis:* ansys.geometry.core.math.vector.Vector3D, *angle: float*) → *Matrix44*

Create a matrix representing a rotation about a given axis.

Parameters

axis
[Vector3D] The axis of rotation.

angle
[float] The angle of rotation in radians.

Returns

Matrix44
A 4x4 matrix representing the rotation.

classmethod Matrix44.**create_matrix_from_mapping**(*frame:* ansys.geometry.core.math.frame.Frame) → *Matrix44*

Create a matrix representing the specified mapping.

Parameters

frame
[Frame] The frame containing the origin and direction vectors.

Returns

Matrix44
A 4x4 matrix representing the translation and rotation defined by the frame.

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.
intersect_interval	Find the intersection of two intervals.

Description

Provides auxiliary math functions for PyAnsys Geometry.

Module detail

misc.**get_two_circle_intersections**(*x0: ansys.geometry.core.typing.Real*, *y0: ansys.geometry.core.typing.Real*, *r0: ansys.geometry.core.typing.Real*, *x1: ansys.geometry.core.typing.Real*, *y1: ansys.geometry.core.typing.Real*, *r1: ansys.geometry.core.typing.Real*) → tuple[tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real], tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real]] | None

Get the intersection points of two circles.

Parameters

x0
[Real] x coordinate of the first circle.

y0
[Real] y coordinate of the first circle.

r0
[Real] Radius of the first circle.

x1
[Real] x coordinate of the second circle.

y1
[Real] y coordinate of the second circle.

r1
[Real] Radius of the second circle.

Returns

tuple[tuple[Real, Real], tuple[Real, Real]] | None
Intersection points of the two circles if they intersect. The points are returned as `((x3, y3), (x4, y4))`, where `(x3, y3)` and `(x4, y4)` are the intersection points of the two circles. If the circles do not intersect, then `None` is returned.

Notes

This function is based on the following StackOverflow post: https://stackoverflow.com/questions/55816902/finding-the-intersection-of-two-circles

That post is based on the following implementation: https://paulbourke.net/geometry/circlesphere/

misc.**intersect_interval**(*first_min*, *second_min*, *first_max*, *second_max*) → tuple[bool, ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real]

Find the intersection of two intervals.

Parameters

first_min
[Real] The minimum value of the first interval.

second_min
[Real] The minimum value of the second interval.

first_max
[Real] The maximum value of the first interval.

second_max
[Real] The maximum value of the second interval.

Returns

tuple[bool, Real, Real]
Tuple with a boolean to indicate whether the intervals intersect, the minimum value of the intersection interval, and the maximum value of the intersection interval. If they do not intersect, then the boolean is False and the other values are 0.

The plane.py module

Summary

Classes

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

unit	Get the unit of the object.
base_unit	Get the base unit of the object.

Properties

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

Attributes

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.**flat**

Method detail

Point2D.**__eq__**(*other:* Point2D) → bool

Equals operator for the Point2D class.

Point2D.**__ne__**(*other:* Point2D) → bool

Not equals operator for the Point2D class.

Point2D.**__add__**(*other:* Point2D | ansys.geometry.core.math.vector.Vector2D) → *Point2D*

Add operation for the Point2D class.

Point2D.**__sub__**(*other:* Point2D) → *Point2D*

Subtraction operation for the Point2D class.

Point2D.**unit**() → pint.Unit

Get the unit of the object.

Point2D.**base_unit**() → pint.Unit

Get the base unit of the object.

Point3D

class ansys.geometry.core.math.point.**Point3D**(*input: numpy.ndarray | ansys.geometry.core.typing.RealSequence = DEFAULT_POINT3D_VALUES*, *unit: pint.Unit | None = None*)

Bases: numpy.ndarray, *ansys.geometry.core.misc.units.PhysicalQuantity*

Provides geometry primitive representation for a 3D point.

Parameters

input
[ndarray | RealSequence, default: DEFAULT_POINT3D_VALUES] Direction arguments, either as a numpy.ndarray class or as a RealSequence.

unit
[Unit | None, default: DEFAULT_UNITS.LENGTH] Units for defining 3D point values. If not specified, the default unit is DEFAULT_UNITS.LENGTH.

Overview

Methods

unit	Get the unit of the object.
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.
position	Get the position of the point as a numpy array.

Attributes

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.

property Point3D.**position**: numpy.ndarray

Get the position of the point as a numpy array.

Attribute detail

Point3D.**flat**

Method detail

Point3D.**__eq__**(*other:* Point3D) → bool

Equals operator for the Point3D class.

Point3D.**__ne__**(*other:* Point3D) → bool

Not equals operator for the Point3D class.

Point3D.**__add__**(*other:* Point3D | ansys.geometry.core.math.vector.Vector3D) → *Point3D*

Add operation for the Point3D class.

Point3D.__sub__(*other:* Point3D | ansys.geometry.core.math.vector.Vector3D) → *Point3D*

Subtraction operation for the Point3D class.

Point3D.**unit**() → pint.Unit

Get the unit of the object.

Point3D.**base_unit**() → pint.Unit

Get the base unit of the object.

Point3D.**transform**(*matrix:* ansys.geometry.core.math.matrix.Matrix44) → *Point3D*

Transform the 3D point with a transformation matrix.

Parameters

matrix
[Matrix44] 4x4 transformation matrix to apply to the point.

Returns

Point3D
New 3D point that is the transformed copy of the original 3D point after applying the transformation matrix.

Notes

Transform the Point3D object by applying the specified 4x4 transformation matrix and return a new Point3D object representing the transformed point.

Description

Provides geometry primitive representation for 2D and 3D points.

Module detail

point.**DEFAULT_POINT2D_VALUES**

Default values for a 2D point.

point.**DEFAULT_POINT3D_VALUES**

Default values for a 3D point.

point.**BASE_UNIT_LENGTH**

Default value for the length of the base unit.

The vector.py module

Summary

Classes

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.
rotate_vector	Rotate a vector around a given axis by a specified angle.
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.**rotate_vector**(*vector:* Vector3D, *angle: ansys.geometry.core.typing.Real | pint.Quantity |* ansys.geometry.core.misc.measurements.Angle) → *Vector3D*

Rotate a vector around a given axis by a specified angle.

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_all_bodies_from_design	Find all the `Body` objects inside a `Design`.
get_bodies_from_ids	Find the `Body` objects inside a `Design` from its ids.
get_components_from_ids	Find the `Component` 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.
get_vertices_from_ids	Find the `Vertex` objects inside a `Design` from its ids.
get_beams_from_ids	Find the `Beam` objects inside a `Design` from its ids.
convert_color_to_hex	Get the hex string color from input formats.
convert_opacity_to_hex	Get the hex string from an opacity value.

Constants

DEFAULT_COLOR

Description

Auxiliary functions for the PyAnsys Geometry library.

Module detail

auxiliary.**get_design_from_component**(*component:* ansys.geometry.core.designer.component.Component) → *ansys.geometry.core.designer.design.Design*

Get the `Design` of the given `Component` object.

Parameters

component
[`Component`] The component object for which to find the `Design`.

Returns

`Design`
The `Design` of the provided component object.

auxiliary.**get_design_from_body**(*body:* ansys.geometry.core.designer.body.Body) → *ansys.geometry.core.designer.design.Design*

Get the `Design` of the given `Body` object.

Parameters

body
[`Body`] The body object for which to find the `Design`.

Returns

`Design`
The `Design` of the provided body object.

auxiliary.**get_design_from_face**(*face:* ansys.geometry.core.designer.face.Face) → *ansys.geometry.core.designer.design.Design*

Get the `Design` of the given `Face` object.

Parameters

face
[`Face`] The face object for which to find the `Design`.

Returns

`Design`
The `Design` of the provided face object.

auxiliary.**get_design_from_edge**(*edge:* ansys.geometry.core.designer.edge.Edge) → *ansys.geometry.core.designer.design.Design*

Get the `Design` of the given `Edge` object.

Parameters

edge
[`Edge`] The edge object for which to find the `Design`.

Returns

`Design`
The `Design` of the provided edge object.

auxiliary.**get_all_bodies_from_design**(*design:* ansys.geometry.core.designer.design.Design) → list[*ansys.geometry.core.designer.body.Body*]

Find all the `Body` objects inside a `Design`.

Parameters

design
[Design] Parent design for the bodies.

Returns

list[Body]
List of Body objects.

Notes

This method takes a design and gets the corresponding Body objects.

auxiliary.**get_bodies_from_ids**(*design:* ansys.geometry.core.designer.design.Design, *body_ids: list[str]*) → list[*ansys.geometry.core.designer.body.Body*]

Find the Body objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the faces.

body_ids
[list[str]] List of body ids.

Returns

list[Body]
List of Body objects.

Notes

This method takes a design and body ids, and gets their corresponding Body object.

auxiliary.**get_components_from_ids**(*design:* ansys.geometry.core.designer.design.Design, *component_ids: list[str]*) → list[*ansys.geometry.core.designer.component.Component*]

Find the Component objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the components.

component_ids
[list[str]] List of component ids.

Returns

list[Component]
List of Component objects.

Notes

This method takes a design and component ids, and gets their corresponding Component object.

auxiliary.**get_faces_from_ids**(*design:* ansys.geometry.core.designer.design.Design, *face_ids: list[str]*) → list[*ansys.geometry.core.designer.face.Face*]

Find the Face objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the faces.

face_ids
[list[str]] List of face ids.

Returns

list[Face]
List of Face objects.

Notes

This method takes a design and face ids, and gets their corresponding `Face` object.

auxiliary.**get_edges_from_ids**(*design:* ansys.geometry.core.designer.design.Design, *edge_ids: list[str]*) → list[*ansys.geometry.core.designer.edge.Edge*]

Find the `Edge` objects inside a `Design` from its ids.

Parameters

design
[Design] Parent design for the edges.

edge_ids
[list[str]] List of edge ids.

Returns

list[Edge]
List of Edge objects.

Notes

This method takes a design and edge ids, and gets their corresponding `Edge` objects.

auxiliary.**get_vertices_from_ids**(*design:* ansys.geometry.core.designer.design.Design, *vertex_ids: list[str]*) → list[*ansys.geometry.core.designer.vertex.Vertex*]

Find the `Vertex` objects inside a `Design` from its ids.

Parameters

design
[Design] Parent design for the vertices.

vertex_ids
[list[str]] List of vertex ids.

Returns

list[Vertex]
List of Vertex objects.

Notes

This method takes a design and vertex ids, and gets their corresponding `Vertex` objects.

auxiliary.**get_beams_from_ids**(*design:* ansys.geometry.core.designer.design.Design, *beam_ids: list[str]*) → list[*ansys.geometry.core.designer.beam.Beam*]

Find the `Beam` objects inside a `Design` from its ids.

Parameters

design
[Design] Parent design for the beams.

beam_ids
[list[str]] List of beam ids.

Returns

list[Beam]
List of Beam objects.

Notes

This method takes a design and beam ids, and gets their corresponding Beam objects.

auxiliary.**convert_color_to_hex**(*color: str | tuple[float, float, float] | tuple[float, float, float, float]*) → str

Get the hex string color from input formats.

Parameters

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

Returns

str
The hex code string for the color, formatted #rrggbbaa.

auxiliary.**convert_opacity_to_hex**(*opacity: float*) → str

Get the hex string from an opacity value.

Parameters

opacity
[float] Opacity to set body to. Must be in the range [0, 1].

Returns

The hex code for the opacity formatted #aa

auxiliary.**DEFAULT_COLOR = '#D6F7D1'**

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.
run_if_graphics_required	Check if graphics are available.
graphics_required	Decorate a method as requiring graphics.

Constants

ERROR_GRAPHICS_REQUIRED	Message to display when graphics are required for a method.

Description

Provides functions for performing common checks.

Module detail

checks.**ensure_design_is_active**(*method*)

Make sure that the design is active before executing a method.

This function is necessary to be called whenever we do any operation on the design. If we are just accessing information of the class, it is not necessary to call this.

checks.**check_is_float_int**(*param: object*, *param_name: str | None = None*) → None

Check if a parameter has a float or integer value.

Parameters

param
[`object`] Object instance to check.

param_name
[`str`, default: `None`] Parameter name (if any).

Raises

`TypeError`
If the parameter does not have a float or integer value.

checks.**check_ndarray_is_float_int**(*param: numpy.ndarray*, *param_name: str | None = None*) → None

Check if a `numpy.ndarray` has float/integer types.

Parameters

param
[ndarray] numpy.ndarray instance to check.

param_name
[str, default: None] numpy.ndarray instance name (if any).

Raises

TypeError
If the numpy.ndarray instance does not have float or integer values.

checks.**check_ndarray_is_not_none**(*param: numpy.ndarray*, *param_name: str | None = None*) → None

Check if a numpy.ndarray is all None.

Parameters

param
[ndarray] numpy.ndarray instance to check.

param_name
[str, default: None] numpy.ndarray instance name (if any).

Raises

ValueError
If the numpy.ndarray instance has a value of None for all parameters.

checks.**check_ndarray_is_all_nan**(*param: numpy.ndarray*, *param_name: str | None = None*) → None

Check if a numpy.ndarray is all nan-valued.

Parameters

param
[ndarray] numpy.ndarray instance to check.

param_name
[str or None, default: None] numpy.ndarray instance name (if any).

Raises

ValueError
If the numpy.ndarray instance is all nan-valued.

checks.**check_ndarray_is_non_zero**(*param: numpy.ndarray*, *param_name: str | None = None*) → None

Check if a numpy.ndarray is zero-valued.

Parameters

param
[ndarray] numpy.ndarray instance to check.

param_name
[str, default: None] numpy.ndarray instance name (if any).

Raises

ValueError
If the numpy.ndarray instance is zero-valued.

checks.**check_pint_unit_compatibility**(*input: pint.Unit*, *expected: pint.Unit*) → None

Check if input pint.Unit is compatible with the expected input.

Parameters

input
[Unit] pint.Unit input.

expected
[Unit] pint.Unit expected dimensionality.

Raises

TypeError
If the input is not compatible with the pint.Unit class.

checks.**check_type_equivalence**(*input: object*, *expected: object*) → None

Check if an input object is of the same class as an expected object.

Parameters

input
[object] Input object.

expected
[object] Expected object.

Raises

TypeError
If the objects are not of the same class.

checks.**check_type**(*input: object*, *expected_type: type | tuple[type, Ellipsis]*) → None

Check if an input object is of the same type as expected types.

Parameters

input
[object] Input object.

expected_type
[type | tuple[type, ...]] One or more types to compare the input object against.

Raises

TypeError
If the object does not match the one or more expected types.

checks.**check_type_all_elements_in_iterable**(*input: collections.abc.Iterable*, *expected_type: type | tuple[type, Ellipsis]*) → None

Check if all elements in an iterable are of the same type as expected.

Parameters

input
[Iterable] Input iterable.

expected_type
[type | tuple[type, ...]] One or more types to compare the input object against.

Raises

TypeError
If one of the elements in the iterable does not match the one or more expected types.

checks.**min_backend_version**(*major: int*, *minor: int*, *service_pack: int*)

Compare a minimum required version to the current backend version.

Parameters

major
[int] Minimum major version required by the method.

minor
[int] Minimum minor version required by the method.

service_pack
[int] Minimum service pack version required by the method.

Raises

GeometryRuntimeError
If the method version is higher than the backend version.

GeometryRuntimeError
If the client is not available.

checks.**deprecated_method**(*alternative: str | None = None*, *info: str | None = None*, *version: str | None = None*, *remove: str | None = None*)

Decorate a method as deprecated.

Parameters

alternative
[str, default: None] Alternative method to use. If provided, the warning message will include the alternative method.

info
[str, default: None] Additional information to include in the warning message.

version
[str, default: None] Version where the method was deprecated.

remove
[str, default: None] Version where the method will be removed.

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

Decorate a method argument as deprecated.

Parameters

arg
[str] Argument to mark as deprecated.

alternative
[str, default: None] Alternative argument to use. If provided, the warning message will include the alternative argument.

info
[str, default: None] Additional information to include in the warning message.

version
[str, default: None] Version where the method was deprecated.

remove
[str, default: None] Version where the method will be removed.

checks.**run_if_graphics_required**()

Check if graphics are available.

checks.**graphics_required**(*method*)

Decorate a method as requiring graphics.

Parameters

method
[callable()] Method to decorate.

Returns

callable()
Decorated method.

checks.**ERROR_GRAPHICS_REQUIRED = 'Graphics are required for this method. Please install the ``graphics`` target to use this...**

Message to display when graphics are required for a method.

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.
Area	Provides the Measurement subclass for holding an area.

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.
AREA	Default area 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.**AREA: pint.Unit**

Default area 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

__repr__	Representation of the `Measurement` class.
__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.**__repr__**()

Representation of the Measurement class.

Measurement.**__eq__**(*other:* Measurement) → bool

Equals operator for the Measurement class.

Distance

class ansys.geometry.core.misc.measurements.**Distance**(*value: ansys.geometry.core.typing.Real | pint.Quantity*, *unit: pint.Unit | None = None*)

Bases: Measurement

Provides the Measurement subclass for holding a distance.

Parameters

value
[Real | Quantity] Value of the distance.

unit
[Unit, default: DEFAULT_UNITS.LENGTH] Units for the distance.

Import detail

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

Angle

class ansys.geometry.core.misc.measurements.**Angle**(*value: ansys.geometry.core.typing.Real | pint.Quantity*, *unit: pint.Unit | None = None*)

Bases: Measurement

Provides the Measurement subclass for holding an angle.

Parameters

value
[Real | Quantity] Value of the angle.

unit
[Unit, default: DEFAULT_UNITS.ANGLE] Units for the distance.

Import detail

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

Area

class ansys.geometry.core.misc.measurements.**Area**(*value: ansys.geometry.core.typing.Real | pint.Quantity*, *unit: pint.Unit | None = None*)

Bases: Measurement

Provides the Measurement subclass for holding an area.

Parameters

value
[Real | Quantity] Value of the area.

unit
[Unit, default: DEFAULT_UNITS.AREA] Units for the area.

Import detail

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

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.
TessellationOptions	Provides options for getting tessellation.

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.

TessellationOptions

class ansys.geometry.core.misc.options.**TessellationOptions**(*surface_deviation: ansys.geometry.core.typing.Real*, *angle_deviation: ansys.geometry.core.typing.Real*, *max_aspect_ratio: ansys.geometry.core.typing.Real = 0.0*, *max_edge_length: ansys.geometry.core.typing.Real = 0.0*, *watertight: bool = False*)

Provides options for getting tessellation.

Parameters

surface_deviation
[Real] The maximum deviation from the true surface position.

angle_deviation
[Real] The maximum deviation from the true surface normal, in radians.

max_aspect_ratio
[Real, default=0.0] The maximum aspect ratio of facets.

max_edge_length
[Real, default=0.0] The maximum facet edge length.

watertight
[bool, default=False] Whether triangles on opposite sides of an edge should match.

Overview

Properties

surface_deviation	Surface Deviation.
angle_deviation	Angle deviation.
max_aspect_ratio	Maximum aspect ratio.
max_edge_length	Maximum edge length.
watertight	Watertight.

Import detail

```
from ansys.geometry.core.misc.options import TessellationOptions
```

Property detail

property TessellationOptions.**surface_deviation: ansys.geometry.core.typing.Real**

Surface Deviation.

The maximum deviation from the true surface position.

property TessellationOptions.**angle_deviation: ansys.geometry.core.typing.Real**

Angle deviation.

The maximum deviation from the true surface normal, in radians.

property TessellationOptions.**max_aspect_ratio: ansys.geometry.core.typing.Real**

Maximum aspect ratio.

The maximum aspect ratio of facets.

property TessellationOptions.**max_edge_length: ansys.geometry.core.typing.Real**

Maximum edge length.

The maximum facet edge length.

property TessellationOptions.**watertight: bool**

Watertight.

Whether triangles on opposite sides of an edge should match.

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 parameters package

Summary

Submodules

parameter	Provides get and set methods for parameters.

The parameter.py module

Summary

Classes

Parameter	Represents a parameter.

Enums

ParameterType	Provides values for the parameter types supported.
ParameterUpdateStatus	Provides values for the status messages associated with parameter updates.

Parameter

class ansys.geometry.core.parameters.parameter.**Parameter**(*id: int*, *name: str*, *dimension_type: ParameterType*, *dimension_value: ansys.geometry.core.typing.Real*)

Represents a parameter.

Parameters

id
[int] Unique ID for the parameter.

name
[str] Name of the parameter.

dimension_type
[ParameterType] Type of the parameter.

dimension_value
[float] Value of the parameter.

Overview

Properties

name	Get the name of the parameter.
dimension_value	Get the value of the parameter.
dimension_type	Get the type of the parameter.

Attributes

id

Import detail

```
from ansys.geometry.core.parameters.parameter import Parameter
```

Property detail

property Parameter.**name: str**

Get the name of the parameter.

property Parameter.**dimension_value: ansys.geometry.core.typing.Real**

Get the value of the parameter.

property Parameter.**dimension_type: *ParameterType***

Get the type of the parameter.

Attribute detail

Parameter.**id**

ParameterType

class ansys.geometry.core.parameters.parameter.**ParameterType**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for the parameter types supported.

Overview

Attributes

DIMENSIONTYPE_UNKNOWN
DIMENSIONTYPE_LINEAR
DIMENSIONTYPE_DIAMETRIC
DIMENSIONTYPE_RADIAL
DIMENSIONTYPE_ARC
DIMENSIONTYPE_AREA
DIMENSIONTYPE_VOLUME
DIMENSIONTYPE_MASS
DIMENSIONTYPE_ANGULAR
DIMENSIONTYPE_COUNT
DIMENSIONTYPE_UNITLESS

Import detail

```
from ansys.geometry.core.parameters.parameter import ParameterType
```

Attribute detail

ParameterType.**DIMENSIONTYPE_UNKNOWN = 0**

ParameterType.**DIMENSIONTYPE_LINEAR = 1**

ParameterType.**DIMENSIONTYPE_DIAMETRIC = 2**

ParameterType.**DIMENSIONTYPE_RADIAL = 3**

ParameterType.**DIMENSIONTYPE_ARC = 4**

ParameterType.**DIMENSIONTYPE_AREA = 5**

ParameterType.**DIMENSIONTYPE_VOLUME = 6**

ParameterType.**DIMENSIONTYPE_MASS = 7**

ParameterType.**DIMENSIONTYPE_ANGULAR = 8**

ParameterType.**DIMENSIONTYPE_COUNT = 9**

ParameterType.**DIMENSIONTYPE_UNITLESS = 10**

ParameterUpdateStatus

class ansys.geometry.core.parameters.parameter.**ParameterUpdateStatus**(**args*, ***kwds*)

Bases: enum.Enum

Provides values for the status messages associated with parameter updates.

Overview

Attributes

SUCCESS
FAILURE
CONSTRAINED_PARAMETERS
UNKNOWN

Import detail

```
from ansys.geometry.core.parameters.parameter import ParameterUpdateStatus
```

Attribute detail

ParameterUpdateStatus.**SUCCESS = 0**

ParameterUpdateStatus.**FAILURE = 1**

ParameterUpdateStatus.**CONSTRAINED_PARAMETERS = 2**

ParameterUpdateStatus.**UNKNOWN = 3**

Description

Provides get and set methods for parameters.

Description

PyAnsys Geometry parameters subpackage.

The plotting package

Summary

Subpackages

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: ansys.tools.visualization_interface.l*

Bases: ansys.tools.visualization_interface.backends.pyvista.widgets.PlotterWidget

Provides the a button to hide/show DesignPoint objects in the plotter.

Parameters

plotter_helper
[GeometryPlotter] Provides the plotter to add the button to.

Overview

Methods

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.

Constants

POLYDATA_COLOR_CYCLER

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_face	Add a face 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.
export_glb	Export the design to a glb file. Does not support picked objects.

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_face**(*face:* ansys.geometry.core.designer.face.Face, ***plotting_options: dict | None*) → None

Add a face to the scene.

Parameters

face
[Face] Face to add.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the Plotter.add_mesh method.

GeometryPlotter.**add_component**(*component:* ansys.geometry.core.designer.component.Component, *merge_component: bool = False*, *merge_bodies: bool = False*, ***plotting_options*) → None

Add a component to the scene.

Parameters

component
[Component] Component to add.

merge_component
[bool, default: False] Whether to merge the component into a single dataset. When True, all the individual bodies are effectively combined into a single dataset without any hierarchy.

merge_bodies
[bool, default: False] Whether to merge each body into a single dataset. When True, all the faces of each individual body are effectively combined into a single dataset without separating faces.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the Plotter.add_mesh method.

GeometryPlotter.**add_component_by_body**(*component:* ansys.geometry.core.designer.component.Component, *merge_bodies: bool*, ***plotting_options: dict | None*) → None

Add a component on a per body basis.

Parameters

component
[Component] Component to add.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the Plotter.add_mesh method.

Notes

This will allow to make use of the service colors. At the same time, it will be slower than the add_component method.

GeometryPlotter.**add_sketch_polydata**(*polydata_entries: list[pyvista.PolyData]*, *sketch:* ansys.geometry.core.sketch.sketch.Sketch *= None*, ***plotting_options*) → None

Add sketches to the scene from PyVista polydata.

Parameters

polydata_entries
[list[pyvista.PolyData]] Polydata to add.

sketch
[Sketch, default: None] Sketch to add.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the Plotter.add_mesh method.

GeometryPlotter.**add_design_point**(*design_point:* ansys.geometry.core.designer.designpoint.DesignPoint, ***plotting_options*) → None

Add a DesignPoint object to the plotter.

Parameters

design_point
[DesignPoint] DesignPoint to add.

GeometryPlotter.**plot_iter**(*plotting_list: list[Any]*, *name_filter: str = None*, ***plotting_options*) → None

Add a list of any type of object to the scene.

These types of objects are supported: Body, Component, list[pv.PolyData], pv.MultiBlock, and Sketch.

Parameters

plotting_list
[list[Any]] list of objects you want to plot.

name_filter
[str, default: None] Regular expression with the desired name or names you want to include in the plotter.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the Plotter.add_mesh method.

GeometryPlotter.**plot**(*plottable_object: Any*, *name_filter: str = None*, ***plotting_options*) → None

Add a custom mesh to the plotter.

Parameters

plottable_object
[str, default: None] Regular expression with the desired name or names you want to include in the plotter.

name_filter: str, default: None
Regular expression with the desired name or names you want to include in the plotter.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, depend of the backend implementation you are using.

GeometryPlotter.**show**(*plotting_object: Any = None*, *screenshot: str | None = None*, ***plotting_options*) → None | list[Any]

Show the plotter.

Parameters

plotting_object
[Any, default: None] Object you can add to the plotter.

screenshot
[str, default: None] Path to save a screenshot of the plotter.

****plotting_options**
[dict, default: None] Keyword arguments for the plotter. Arguments depend of the backend implementation you are using.

GeometryPlotter.**export_glb**(*plotting_object: Any = None*, *filename: str | pathlib.Path | None = None*, ***plotting_options*) → pathlib.Path

Export the design to a glb file. Does not support picked objects.

Parameters

plotting_object
[Any, default: None] Object you can add to the plotter.

filename
[str | Path, default: None] Path to save a GLB file of the plotter. If None, the file will be saved as temp_glb.glb.

****plotting_options**
[dict, default: None] Keyword arguments for the plotter. Arguments depend of the backend implementation you are using.

Returns

Path
Path to the exported glb file.

Description

Provides plotting for various PyAnsys Geometry objects.

Module detail

plotter.**POLYDATA_COLOR_CYCLER**

Description

Provides the PyAnsys Geometry plotting subpackage.

The shapes package

Summary

Subpackages

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.
nurbs	Provides for creating and managing a NURBS curve.
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 evaluate 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.

abstractmethod Circle.**contains_param**(*param: ansys.geometry.core.typing.Real*) → bool

Check a parameter is within the parametric range of the curve.

abstractmethod 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

abstractmethod Curve.**parameterization**() → *ansys.geometry.core.shapes.parameterization.Parameterization*

Parameterize the curve.

abstractmethod Curve.**contains_param**(*param: ansys.geometry.core.typing.Real*) → bool

Check a parameter is within the parametric range of the curve.

abstractmethod 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.

abstractmethod Curve.**transformed_copy**(*matrix:* ansys.geometry.core.math.matrix.Matrix44) → *Curve*

Create a transformed copy of the curve.

abstractmethod Curve.**__eq__**(*other:* Curve) → bool

Determine if two curves are equal.

abstractmethod Curve.**evaluate**(*parameter: ansys.geometry.core.typing.Real*) → *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Evaluate the curve at the given parameter.

abstractmethod 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.

abstractmethod Ellipse.**contains_param**(*param: ansys.geometry.core.typing.Real*) → bool

Check a parameter is within the parametric range of the curve.

abstractmethod 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

```
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.

abstractmethod `Line.contains_param(param: ansys.geometry.core.typing.Real)` → bool

Check a parameter is within the parametric range of the curve.

abstractmethod `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 `nurbs.py` module

Summary

Classes

NURBSCurve	Represents a NURBS curve.
NURBSCurveEvaluation	Provides evaluation of a NURBS curve at a given parameter.

NURBSCurve

class ansys.geometry.core.shapes.curves.nurbs.**NURBSCurve**

Bases: *ansys.geometry.core.shapes.curves.curve.Curve*

Represents a NURBS curve.

Notes

This class is a wrapper around the NURBS curve class from the *geomdl* library. By leveraging the *geomdl* library, this class provides a high-level interface to create and manipulate NURBS curves. The *geomdl* library is a powerful library for working with NURBS curves and surfaces. For more information, see https://pypi.org/project/geomdl/.

Overview

Abstract methods

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

Constructors

from_control_points	Create a NURBS curve from control points.
fit_curve_from_points	Fit a NURBS curve to a set of points.

Methods

parameterization	Get the parametrization of the NURBS curve.
transformed_copy	Create a transformed copy of the curve.
evaluate	Evaluate the curve at the given parameter.
project_point	Project a point to the NURBS curve.

Properties

geomdl_nurbs_curve	Get the underlying NURBS curve.
control_points	Get the control points of the curve.
degree	Get the degree of the curve.
knots	Get the knot vector of the curve.
weights	Get the weights of the control points.

Special methods

__eq__	Determine if two curves are equal.

Import detail

```
from ansys.geometry.core.shapes.curves.nurbs import NURBSCurve
```

Property detail

property NURBSCurve.**geomdl_nurbs_curve: geomdl.NURBS.Curve**

Get the underlying NURBS curve.

Notes

This property gives access to the full functionality of the NURBS curve coming from the *geomdl* library. Use with caution.

property NURBSCurve.**control_points: list[ansys.geometry.core.math.Point3D]**

Get the control points of the curve.

property NURBSCurve.**degree: int**

Get the degree of the curve.

property NURBSCurve.**knots: list[ansys.geometry.core.typing.Real]**

Get the knot vector of the curve.

property NURBSCurve.**weights: list[ansys.geometry.core.typing.Real]**

Get the weights of the control points.

Method detail

classmethod NURBSCurve.**from_control_points**(*control_points: list[ansys.geometry.core.math.Point3D]*, *degree: int*, *knots: list[ansys.geometry.core.typing.Real]*, *weights: list[ansys.geometry.core.typing.Real] = None*) → *NURBSCurve*

Create a NURBS curve from control points.

Parameters

control_points
[list[Point3D]] Control points of the curve.

degree
[int] Degree of the curve.

knots
[list[Real]] Knot vector of the curve.

weights
[list[Real], optional] Weights of the control points.

Returns

NURBSCurve
NURBS curve.

classmethod NURBSCurve.**fit_curve_from_points**(*points: list[ansys.geometry.core.math.Point3D]*, *degree: int*) → *NURBSCurve*

Fit a NURBS curve to a set of points.

Parameters

points
[list[Point3D]] Points to fit the curve to.

degree
[int] Degree of the curve.

Returns

NURBSCurve
Fitted NURBS curve.

NURBSCurve.__eq__(*other:* NURBSCurve) → bool

Determine if two curves are equal.

NURBSCurve.**parameterization**() → *ansys.geometry.core.shapes.parameterization.Parameterization*

Get the parametrization of the NURBS curve.

The parameter is defined in the interval [0, 1] by default. Information is provided about the parameter type and form.

Returns

Parameterization
Information about how the NURBS curve is parameterized.

NURBSCurve.**transformed_copy**(*matrix: ansys.geometry.core.math.Matrix44*) → *NURBSCurve*

Create a transformed copy of the curve.

Parameters

matrix
[Matrix44] Transformation matrix.

Returns

NURBSCurve
Transformed copy of the curve.

NURBSCurve.**evaluate**(*parameter: ansys.geometry.core.typing.Real*) → *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Evaluate the curve at the given parameter.

Parameters

parameter
[Real] Parameter to evaluate the curve at.

Returns

CurveEvaluation
Evaluation of the curve at the given parameter.

abstractmethod NURBSCurve.**contains_param**(*param: ansys.geometry.core.typing.Real*) → bool

Check a parameter is within the parametric range of the curve.

abstractmethod NURBSCurve.**contains_point**(*point: ansys.geometry.core.math.Point3D*) → bool

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

NURBSCurve.**project_point**(*point: ansys.geometry.core.math.Point3D*, *initial_guess: ansys.geometry.core.typing.Real | None = None*) → *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Project a point to the NURBS curve.

This method returns the evaluation at the closest point.

Parameters

point
[Point3D] Point to project to the curve.

initial_guess
[Real, optional] Initial guess for the optimization algorithm. If not provided, the midpoint of the domain is used.

Returns

CurveEvaluation
Evaluation at the closest point on the curve.

Notes

Based on the NURBS book, the projection of a point to a NURBS curve is the solution to the following optimization problem: minimize the distance between the point and the curve. The distance is defined as the Euclidean distance squared. For more information, please refer to the implementation of the *distance_squared* function.

NURBSCurveEvaluation

class ansys.geometry.core.shapes.curves.nurbs.**NURBSCurveEvaluation**(*nurbs_curve: NURBSCurve*, *parameter: ansys.geometry.core.typing.Real*)

Bases: *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Provides evaluation of a NURBS curve at a given parameter.

Parameters

nurbs_curve: ~ansys.geometry.core.shapes.curves.nurbs.NURBSCurve
NURBS curve to evaluate.

parameter: Real
Parameter to evaluate the NURBS curve at.

Overview

Properties

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.nurbs import NURBSCurveEvaluation
```

Property detail

property NURBSCurveEvaluation.**parameter: ansys.geometry.core.typing.Real**

Parameter that the evaluation is based upon.

property NURBSCurveEvaluation.**position: ansys.geometry.core.math.Point3D**

Position of the evaluation.

property NURBSCurveEvaluation.**first_derivative:** *ansys.geometry.core.math.vector.Vector3D*

First derivative of the evaluation.

property NURBSCurveEvaluation.**second_derivative:** *ansys.geometry.core.math.vector.Vector3D*

Second derivative of the evaluation.

property NURBSCurveEvaluation.**curvature: ansys.geometry.core.typing.Real**

Curvature of the evaluation.

Description

Provides for creating and managing a NURBS curve.

The `trimmed_curve.py` module

Summary

Classes

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.
transformed_copy	Return a copy of the trimmed curve transformed by the given matrix.
translate	Translate the trimmed curve by a given vector and distance.
rotate	Rotate the trimmed curve around a given axis centered at a given point.

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.**transformed_copy**(*matrix:* ansys.geometry.core.math.matrix.Matrix44) → *TrimmedCurve*

Return a copy of the trimmed curve transformed by the given matrix.

Parameters

matrix
[`Matrix44`] Transformation matrix to apply to the curve.

Returns

TrimmedCurve
A new trimmed curve with the transformation applied.

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

Translate the trimmed curve by a given vector and distance.

Parameters

direction
[`UnitVector3D`] Direction of translation.

distance
[Real | Quantity | Distance] Distance to translate.

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

Rotate the trimmed curve around a given axis centered at a given point.

Parameters

origin
[Point3D] Origin point of the rotation.

axis
[UnitVector3D] Axis of rotation.

angle
[Real | Quantity | Angle] Angle to rotate in radians.

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.G = *None*)

Bases: TrimmedCurve

Represents a reversed trimmed curve.

When a curve is reversed, its start and end points are swapped, and parameters for evaluations are handled to provide expected results conforming to the direction of the curve. For example, evaluating a trimmed curve proportionally at 0 evaluates at the start point of the curve, but evaluating a reversed trimmed curve proportionally at 0 evaluates at what was previously the end point of the curve but is now the start point.

Parameters

geometry
[Curve] Underlying mathematical representation of the curve.

start
[Point3D] Original start point of the curve.

end
[Point3D] Original end point of the curve.

interval
[Interval] Parametric interval that bounds the curve.

length
[Quantity] Length of the curve.

grpc_client
[GrpcClient, default: None] gRPC client that is required for advanced functions such as *intersect_curve()*.

Overview

Methods

evaluate_proportion	Evaluate the curve at a proportional value.

Import detail

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

Method detail

ReversedTrimmedCurve.**evaluate_proportion**(*param: ansys.geometry.core.typing.Real*) → *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Evaluate the curve at a proportional value.

A parameter of 0 corresponds to the start of the curve, while a parameter of 1 corresponds to the end of the curve.

Parameters

param
[Real] Parameter in the proportional range [0,1].

Returns

CurveEvaluation
Resulting curve evaluation.

Description

Trimmed curve class.

Description

Provides the PyAnsys Geometry curves subpackage.

The surfaces package

Summary

Submodules

cone	Provides for creating and managing a cone.
cylinder	Provides for creating and managing a cylinder.
plane	Provides for creating and managing a plane.
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.

abstractmethod Cone.**contains_param**(*param_uv:* ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod 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.

abstractmethod Cylinder.contains_param(*param_uv:* ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod 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.

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 plane.
dir_x	X-direction of the plane.
dir_y	Y-direction of the plane.
dir_z	Z-direction of the plane.

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 plane.

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

X-direction of the plane.

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

Y-direction of the plane.

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

Z-direction of the plane.

Method detail

PlaneSurface.**__eq__**(*other:* PlaneSurface) → bool

Check whether two planes are equal.

abstractmethod PlaneSurface.**contains_param**(*param_uv:* ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a ParamUV is within the parametric range of the surface.

abstractmethod 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 plane.

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.

abstractmethod Sphere.**contains_param**(*param_uv:* ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod 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

abstractmethod 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).

abstractmethod Surface.**contains_param**(*param_uv:* ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod 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.

abstractmethod Surface.**transformed_copy**(*matrix:* ansys.geometry.core.math.matrix.Matrix44) → *Surface*

Create a transformed copy of the surface.

abstractmethod Surface.**__eq__**(*other:* Surface) → bool

Determine if two surfaces are equal.

abstractmethod Surface.**evaluate**(*parameter:* ansys.geometry.core.shapes.parameterization.ParamUV) → *ansys.geometry.core.shapes.surfaces.surface_evaluation.SurfaceEvaluation*

Evaluate the surface at the given parameter.

abstractmethod 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.

abstractmethod Torus.**contains_param**(*param_uv:* ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod 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:* ansys.geometry.core.shapes.sur
box_uv: ansys.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.

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.

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.

Import detail

```
from ansys.geometry.core.sketch.box import Box
```

Property detail

property Box.**center:** *ansys.geometry.core.math.point.Point2D*

Center point of the box.

property Box.**width:** pint.Quantity

Width of the box.

property Box.**height:** pint.Quantity

Height of the box.

property Box.perimeter: pint.Quantity

Perimeter of the box.

property Box.area: pint.Quantity

Area of the box.

property Box.visualization_polydata: pyvista.PolyData

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global cartesian coordinate system.

Returns

pyvista.PolyData
VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a box (quadrilateral).

The circle.py module

Summary

Classes

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.

Import detail

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

SpurGear

class ansys.geometry.core.sketch.gears.**SpurGear**(*origin:* ansys.geometry.core.math.point.Point2D, *module: ansys.geometry.core.typing.Real*, *pressure_angle: pint.Quantity* | ansys.geometry.core.misc.measurements.Angle | *ansys.geometry.core.typing.Real*, *n_teeth: int*)

Bases: Gear

Provides the class for sketching spur gears.

Parameters

origin
[Point2D] Origin of the spur gear.

module
[Real] Module of the spur gear. This is also the ratio between the pitch circle diameter in millimeters and the number of teeth.

pressure_angle
[Quantity | Angle | Real] Pressure angle of the spur gear.

n_teeth
[int] Number of teeth of the spur gear.

Overview

Properties

origin	Origin of the spur gear.
module	Module of the spur gear.
pressure_angle	Pressure angle 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.**origin: *ansys.geometry.core.math.point.Point2D***

Origin of the spur gear.

property SpurGear.**module: ansys.geometry.core.typing.Real**

Module of the spur gear.

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

Pressure angle of the spur gear.

property SpurGear.**n_teeth: int**

Number of teeth of the spur gear.

property SpurGear.**ref_diameter: ansys.geometry.core.typing.Real**

Reference diameter of the spur gear.

property SpurGear.**base_diameter: ansys.geometry.core.typing.Real**

Base diameter of the spur gear.

property SpurGear.**addendum: ansys.geometry.core.typing.Real**

Addendum of the spur gear.

property SpurGear.**dedendum: ansys.geometry.core.typing.Real**

Dedendum of the spur gear.

property SpurGear.**tip_diameter: ansys.geometry.core.typing.Real**

Tip diameter of the spur gear.

property SpurGear.**root_diameter: ansys.geometry.core.typing.Real**

Root diameter of the spur gear.

Description

Module for creating and managing gears.

The `polygon.py` module

Summary

Classes

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.

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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.
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

```
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. Default units are the expected units, otherwise it will be inconsistent.

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.

Import detail

```
from ansys.geometry.core.sketch.slot import Slot
```

Property detail

property Slot.**center: *ansys.geometry.core.math.point.Point2D***

Center of the slot.

property Slot.**width: pint.Quantity**

Width of the slot.

property Slot.**height: pint.Quantity**

Height of the slot.

property Slot.**perimeter: pint.Quantity**

Perimeter of the slot.

property Slot.**area: pint.Quantity**

Area of the slot.

property Slot.**visualization_polydata: pyvista.PolyData**

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData
VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a slot.

The `trapezoid.py` module

Summary

Classes

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.

Import detail

```
from ansys.geometry.core.sketch.trapezoid import Trapezoid
```

Property detail

property Trapezoid.**center: *ansys.geometry.core.math.point.Point2D***

Center of the trapezoid.

property Trapezoid.**base_width: pint.Quantity**

Width of the trapezoid.

property Trapezoid.**height: pint.Quantity**

Height of the trapezoid.

property Trapezoid.**visualization_polydata: pyvista.PolyData**

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData
VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a trapezoid.

The `triangle.py` module

Summary

Classes

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

check_geometry	Module for repair tool message.
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.
unsupported	Unsupported functions for the PyAnsys Geometry library.

The check_geometry.py module

Summary

Classes

GeometryIssue	Provides return message for the repair tool methods.
InspectResult	Provides the result of the inspect geometry operation.

GeometryIssue

class ansys.geometry.core.tools.check_geometry.**GeometryIssue**(*message_type: str*, *message_id: str*, *message: str*, *edges: list[str]*, *faces: list[str]*)

Provides return message for the repair tool methods.

Overview

Properties

message_type	The type of the message (warning, error, info).
message_id	The identifier for the message.
message	The content of the message.
edges	The List of edges (if any) that are part of the issue.
faces	The List of faces (if any) that are part of the issue.

Import detail

```
from ansys.geometry.core.tools.check_geometry import GeometryIssue
```

Property detail

property GeometryIssue.**message_type: str**

The type of the message (warning, error, info).

property GeometryIssue.**message_id: str**

The identifier for the message.

property GeometryIssue.**message: str**

The content of the message.

property GeometryIssue.**edges: list[str]**

The List of edges (if any) that are part of the issue.

property GeometryIssue.**faces: list[str]**

The List of faces (if any) that are part of the issue.

InspectResult

class ansys.geometry.core.tools.check_geometry.**InspectResult**(*grpc_client:* ansys.geometry.core.connection.client.GrpcClient, *body:* ansys.geometry.core.designer.body.Body, *issues: list[*GeometryIssue*]*)

Provides the result of the inspect geometry operation.

Overview

Methods

repair	Repair the problem area.

Properties

body	The body for which issues are found.
issues	The list of issues for the body.

Import detail

```
from ansys.geometry.core.tools.check_geometry import InspectResult
```

Property detail

property InspectResult.**body: *ansys.geometry.core.designer.body.Body***

The body for which issues are found.

property InspectResult.**issues: list[*GeometryIssue*]**

The list of issues for the body.

Method detail

InspectResult.**repair**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Repair the problem area.

Returns

RepairToolMessage
Message containing created and/or modified bodies.

Description

Module for repair tool message.

The measurement_tools.py module

Summary

Classes

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**(*object1:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge, *object2:* ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge) → *Gap*

Find the gap between two bodies.

Parameters

object1
[Union[Body, Face, Edge]] First object to measure the gap.

object2
[Union[Body, Face, Edge]] Second object to measure the gap.

Returns

Gap
Gap between two bodies.

Warning

This method is only available starting on Ansys release 24R2.

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.
remove_rounds	Remove rounds from geometry.
share_topology	Share topology between the chosen bodies.
enhanced_share_topology	Share topology between the chosen bodies.
find_logos	Detect logos in geometry.
find_and_remove_logos	Detect and remove logos in geometry.

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.

Warning

This method is only available starting on Ansys release 25R1.

PrepareTools.**extract_volume_from_edge_loops**(*sealing_edges: list[ansys.geometry.core.designer.edge.Edge]*, *inside_faces: list[ansys.geometry.core.designer.face.Face] = None*) → list[*ansys.geometry.core.designer.body.Body*]

Extract a volume from input edge loops.

Creates a volume (typically a flow volume) from a list of edge loops that seal the volume. and one or more faces that define the wetted surface (inside faces of the solid).

Parameters

sealing_edges
[list[Edge]] List of faces that seal the volume.

inside_faces
[list[Face], optional] List of faces that define the interior of the solid (not always necessary).

Returns

list[Body]
List of created bodies.

Warning

This method is only available starting on Ansys release 25R1.

PrepareTools.**remove_rounds**(*faces: list[ansys.geometry.core.designer.face.Face]*, *auto_shrink: bool = False*) → bool

Remove rounds from geometry.

Tries to remove rounds from geometry. Faces to be removed are input to the method.

Parameters

round_faces
[list[Face]] List of rounds faces to be removed

auto_shrink
[bool, default: False] Whether to shrink the geometry after removing rounds. Fills in the gaps left by the removed rounds.

Returns

bool
True if successful, False if failed.

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.

Warning

This method is only available starting on Ansys release 24R2.

`PrepareTools.`**`enhanced_share_topology`**(*bodies: list[ansys.geometry.core.designer.body.Body], tol: ansys.geometry.core.typing.Real = 0.0, preserve_instances: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Share topology between the chosen bodies.

Parameters

bodies
[`list`[Body]] List of bodies to share topology between.

tol
[`Real`] Maximum distance between bodies.

preserve_instances
[bool] Whether instances are preserved.

Returns

`RepairToolMessage`
Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

`PrepareTools.`**`find_logos`**(*bodies: list[ansys.geometry.core.designer.body.Body] = None, min_height: ansys.geometry.core.typing.Real = None, max_height: ansys.geometry.core.typing.Real = None*) → *ansys.geometry.core.tools.problem_areas.LogoProblemArea*

Detect logos in geometry.

Detects logos, using a list of bodies if provided. The logos are returned as a list of faces.

Parameters

bodies
[`list`[Body], `optional`] List of bodies where logos should be detected

min_height
[`real`, `optional`] The minimum height when searching for logos

max_height: real, optional
The minimum height when searching for logos

Returns

LogoProblemArea
Problem area with logo faces.

Warning

This method is only available starting on Ansys release 25R2.

PrepareTools.**find_and_remove_logos**(*bodies: list[ansys.geometry.core.designer.body.Body] = None*, *min_height: ansys.geometry.core.typing.Real = None*, *max_height: ansys.geometry.core.typing.Real = None*) → bool

Detect and remove logos in geometry.

Detects and remove logos, using a list of bodies if provided.

Parameters

bodies
[list[Body], optional] List of bodies where logos should be detected and removed.

min_height
[real, optional] The minimum height when searching for logos

max_height: real, optional
The minimum height when searching for logos

Returns

Boolean value indicating whether the operation was successful.

Warning

This method is only available starting on Ansys release 25R2.

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.
UnsimplifiedFaceProblemAre	Represents a unsimplified face problem area with unique identifier and associated faces.
InterferenceProblemAreas	Represents an interference problem area with a unique identifier and associated bodies.
LogoProblemArea	Represents a logo problem area defined by a list of 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

```
from ansys.geometry.core.tools.problem_areas import ProblemArea
```

Property detail

property ProblemArea.**id: str**

The id of the problem area.

Method detail

abstractmethod 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.Face*]*)

Bases: ProblemArea

Provides duplicate face problem area definition.

Represents a duplicate face problem area with unique identifier and associated faces.

Parameters

id
[str] Server-defined ID for the problem area.

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

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

Overview

Methods

fix	Fix the problem area.

Properties

faces	The list of faces connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import DuplicateFaceProblemAreas
```

Property detail

property DuplicateFaceProblemAreas.**faces: list[*ansys.geometry.core.designer.face.Face*]**

The list of faces connected to this problem area.

Method detail

DuplicateFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

MissingFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.**MissingFaceProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *edges: list[*ansys.geometry.core.designer.edge.Edge

Bases: ProblemArea

Provides missing face problem area definition.

Parameters

id
[str] Server-defined ID for the problem area.

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

edges
[list[Edge]] List of edges associated with the design.

Overview

Methods

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 MissingFaceProblemAreas
```

Property detail

property MissingFaceProblemAreas.**edges: list[*ansys.geometry.core.designer.edge.Edge*]**

The list of edges connected to this problem area.

Method detail

MissingFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

InexactEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**InexactEdgeProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *edges: list[*ansys.geometry.core.designer.edge.Edge

Bases: ProblemArea

Represents an inexact edge problem area with unique identifier and associated edges.

Parameters

id
[str] Server-defined ID for the problem area.

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

edges
[list[Edge]] List of edges associated with the design.

Overview

Methods

fix	Fix the problem area.

Properties

edges	The list of edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import InexactEdgeProblemAreas
```

Property detail

property InexactEdgeProblemAreas.**edges: list[***ansys.geometry.core.designer.edge.Edge***]**

The list of edges connected to this problem area.

Method detail

InexactEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

ExtraEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**ExtraEdgeProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *edges: list[*ansys.geometry.core.designer.edge.Edge*]*)

Bases: ProblemArea

Represents a extra edge problem area with unique identifier and associated edges.

Parameters

id
[str] Server-defined ID for the problem area.

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

edges
[list[Edge]] List of edges associated with the design.

Overview

Methods

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 ExtraEdgeProblemAreas
```

Property detail

property ExtraEdgeProblemAreas.**edges: list[***ansys.geometry.core.designer.edge.Edge***]**

The list of edges connected to this problem area.

Method detail

ExtraEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

ShortEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**ShortEdgeProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *edges: list[*ansys.geometry.core.designer.edge.Edge*]*)

Bases: ProblemArea

Represents a short edge problem area with a unique identifier and associated edges.

Parameters

id
[str] Server-defined ID for the problem area.

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

edges
[list[Edge]] List of edges associated with the design.

Overview

Methods

fix	Fix the problem area.

Properties

edges	The list of edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import ShortEdgeProblemAreas
```

Property detail

property ShortEdgeProblemAreas.**edges: list[***ansys.geometry.core.designer.edge.Edge***]**

The list of edges connected to this problem area.

Method detail

ShortEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

SmallFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.**SmallFaceProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *faces: list[*ansys.geometry.core.designer.face.Face*]*)

Bases: ProblemArea

Represents a small face problem area with a unique identifier and associated faces.

Parameters

id
[str] Server-defined ID for the problem area.

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

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

Overview

Methods

fix	Fix the problem area.

Properties

faces	The list of faces connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import SmallFaceProblemAreas
```

Property detail

property SmallFaceProblemAreas.**faces: list[***ansys.geometry.core.designer.face.Face***]**

The list of faces connected to this problem area.

Method detail

SmallFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

SplitEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**SplitEdgeProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *edges: list[*ansys.geometry.core.designer.edge.Edge*]*)

Bases: ProblemArea

Represents a split edge problem area with unique identifier and associated edges.

Parameters

id
[str] Server-defined ID for the problem area.

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

edges
[list[Edge]] List of edges associated with the design.

Overview

Methods

fix	Fix the problem area.

Properties

edges	The list of edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import SplitEdgeProblemAreas
```

Property detail

property SplitEdgeProblemAreas.**edges: list[*ansys.geometry.core.designer.edge.Edge*]**

The list of edges connected to this problem area.

Method detail

SplitEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

StitchFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.**StitchFaceProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *bodies: list[*ansys.geometry.core.designer.body.Body

Bases: ProblemArea

Represents a stitch face problem area with unique identifier and associated faces.

Parameters

id
[str] Server-defined ID for the problem area.

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

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

Overview

Methods

fix	Fix the problem area.

Properties

bodies	The list of bodies connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import StitchFaceProblemAreas
```

Property detail

property StitchFaceProblemAreas.**bodies: list[***ansys.geometry.core.designer.body.Body***]**

The list of bodies connected to this problem area.

Method detail

StitchFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

UnsimplifiedFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.**UnsimplifiedFaceProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcC*, *faces: list[*ansys.geometry.core.designer.fac

Bases: ProblemArea

Represents a unsimplified face problem area with unique identifier and associated faces.

Parameters

id
[str] Server-defined ID for the problem area.

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

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

Overview

Methods

fix	Fix the problem area.

Properties

faces	The list of faces connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import UnsimplifiedFaceProblemAreas
```

Property detail

property UnsimplifiedFaceProblemAreas.**faces: list[***ansys.geometry.core.designer.face.Face***]**

The list of faces connected to this problem area.

Method detail

UnsimplifiedFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

InterferenceProblemAreas

class ansys.geometry.core.tools.problem_areas.**InterferenceProblemAreas**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *bodies: list[*ansys.geometry.core.designer.body.Bo

Bases: ProblemArea

Represents an interference problem area with a unique identifier and associated bodies.

Parameters

id
[str] Server-defined ID for the problem area.

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

bodies
[list[Body]] List of bodies in the problem area.

Overview

Methods

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 InterferenceProblemAreas
```

Property detail

property InterferenceProblemAreas.**bodies: list[***ansys.geometry.core.designer.body.Body***]**

The list of the bodies connected to this problem area.

Method detail

InterferenceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

Notes

The current implementation does not properly track changes. The list of created and modified bodies are empty.

LogoProblemArea

class ansys.geometry.core.tools.problem_areas.**LogoProblemArea**(*id: str*, *grpc_client: ansys.geometry.core.connection.GrpcClient*, *face_ids: list[str]*)

Bases: ProblemArea

Represents a logo problem area defined by a list of faces.

Parameters

id
[str] Server-defined ID for the problem area.

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

faces
[list[str]] List of faces defining the logo problem area.

Overview

Methods

fix	Fix the problem area by deleting the logos.

Properties

face_ids	The ids of the faces defining the logos.

Import detail

```
from ansys.geometry.core.tools.problem_areas import LogoProblemArea
```

Property detail

property LogoProblemArea.**face_ids: list[str]**

The ids of the faces defining the logos.

Method detail

LogoProblemArea.**fix**() → bool

Fix the problem area by deleting the logos.

Returns

message: bool
Message that return whether the operation was successful.

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]*, *deleted_bodies: list[str] = None*, *craeted_components: list[str] = None*, *modified_components: list[str] = None*, *deleted_components: list[str] = None*, *found: int = -1*, *repaired: int = -1*)

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.
found	Number of problem areas found for the repair operation.
repaired	Number of problem areas repaired during the repair operation.

Import detail

```
from ansys.geometry.core.tools.repair_tool_message import RepairToolMessage
```

Property detail

property RepairToolMessage.**success: bool**

The success of the repair operation.

property RepairToolMessage.**created_bodies: list[str]**

The list of the created bodies after the repair operation.

property RepairToolMessage.**modified_bodies: list[str]**

The list of the modified bodies after the repair operation.

property RepairToolMessage.**found: int**

Number of problem areas found for the repair operation.

property RepairToolMessage.**repaired: int**

Number of problem areas repaired during the repair operation.

Description

Module for repair tool message.

The `repair_tools.py` module

Summary

Classes

RepairTools	Repair tools for PyAnsys Geometry.

RepairTools

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

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.
find_simplify	Detect faces in a body that can be simplified.
find_interferences	Find the interference problem areas.
find_and_fix_short_edges	Find and fix the short edge problem areas.
find_and_fix_extra_edges	Find and fix the extra edge problem areas.
find_and_fix_split_edges	Find and fix the split edge problem areas.
find_and_fix_simplify	Find and simplify the provided geometry.
find_and_fix_stitch_faces	Find and fix the stitch face problem areas.
inspect_geometry	Return a list of geometry issues organized by body.
repair_geometry	Attempt to repair the geometry for the given bodies.

Import detail

```
from ansys.geometry.core.tools.repair_tools import RepairTools
```

Method detail

RepairTools.**find_split_edges**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*, *angle: ansys.geometry.core.typing.Real = 0.0*, *length: ansys.geometry.core.typing.Real = 0.0*) → list[*ansys.geometry.core.tools.problem_areas.SplitEdgeProblemAreas*]

Find split edges in the given list of bodies.

This method finds the split edge problem areas and returns a list of split edge problem areas objects.

Parameters

bodies
[list[Body]] List of bodies that split edges are investigated on.

angle
[Real] The maximum angle between edges.

length
[Real] The maximum length of the edges.

Returns

list[SplitEdgeProblemAreas]
List of objects representing split edge problem areas.

RepairTools.**find_extra_edges**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*) → list[*ansys.geometry.core.tools.problem_areas.ExtraEdgeProblemAreas*]

Find the extra edges in the given list of bodies.

This method find the extra edge problem areas and returns a list of extra edge problem areas objects.

Parameters

bodies
[list[Body]] List of bodies that extra edges are investigated on.

Returns

list[ExtraEdgeProblemArea]
List of objects representing extra edge problem areas.

RepairTools.**find_inexact_edges**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*) → list[*ansys.geometry.core.tools.problem_areas.InexactEdgeProblemAreas*]

Find inexact edges in the given list of bodies.

This method find the inexact edge problem areas and returns a list of inexact edge problem areas objects.

Parameters

bodies
[list[Body]] List of bodies that inexact edges are investigated on.

Returns

list[InExactEdgeProblemArea]
List of objects representing inexact edge problem areas.

RepairTools.**find_short_edges**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*, *length: ansys.geometry.core.typing.Real = 0.0*) → list[*ansys.geometry.core.tools.problem_areas.ShortEdgeProblemAreas*]

Find the short edge problem areas.

This method finds the short edge problem areas and returns a list of these objects.

Parameters

bodies
[list[Body]] List of bodies that short edges are investigated on.

Returns

list[ShortEdgeProblemAreas]
List of objects representing short edge problem areas.

RepairTools.**find_duplicate_faces**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*) → list[*ansys.geometry.core.tools.problem_areas.DuplicateFaceProblemAreas*]

Find the duplicate face problem areas.

This method finds the duplicate face problem areas and returns a list of duplicate face problem areas objects.

Parameters

bodies
[list[Body]] List of bodies that duplicate faces are investigated on.

Returns

list[DuplicateFaceProblemAreas]
List of objects representing duplicate face problem areas.

RepairTools.**find_missing_faces**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*, *angle:* ansys.geometry.core.misc.measurements.Angle | *pint.Quantity* | *ansys.geometry.core.typing.Real* | *None* = *None*, *distance:* ansys.geometry.core.misc.measurements.Distance | *pint.Quantity* | *ansys.geometry.core.typing.Real* | *None* = *None*) → 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.

angle
[Angle | Quantity | Real, optional] The minimum angle between faces. By default, None. This option is only used if the backend version is 26.1 or higher.

distance
[Distance | Quantity | Real, optional] The minimum distance between faces. By default, None. This option is only used if the backend version is 26.1 or higher.

Returns

list[MissingFaceProblemAreas]
List of objects representing missing face problem areas.

RepairTools.**find_small_faces**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*, *area:* ansys.geometry.core.misc.measurements.Area | *pint.Quantity* | *ansys.geometry.core.typing.Real* | *None* = *None*, *width:* ansys.geometry.core.misc.measurements.Distance | *pint.Quantity* | *ansys.geometry.core.typing.Real* | *None* = *None*) → 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.

area
[Area | Quantity | Real, optional] Maximum area of the faces. By default, None. This option is only used if the backend version is 26.1 or higher.

width
[Distance | Quantity | Real, optional] Maximum width of the faces. By default, None. This option is only used if the backend version is 26.1 or higher.

Returns

list[SmallFaceProblemAreas]
List of objects representing small face problem areas.

RepairTools.**find_stitch_faces**(*bodies: list[*ansys.geometry.core.designer.body.Body*]*, *max_distance:* ansys.geometry.core.misc.measurements.Distance | *pint.Quantity* | *ansys.geometry.core.typing.Real* | *None* = *None*) → 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.

max_distance
[Distance | Quantity | Real, optional] Maximum distance between faces. By default, None. This option is only used if the backend version is 26.1 or higher.

Returns

list[StitchFaceProblemAreas]
List of objects representing stitch face problem areas.

RepairTools.**find_simplify**(*bodies: list[ansys.geometry.core.designer.body.Body]*) → list[*ansys.geometry.core.tools.problem_areas.UnsimplifiedFaceProblemAreas*]

Detect faces in a body that can be simplified.

Parameters

bodies
[list[Body]] List of bodies to search.

Returns

list[UnsimplifiedFaceProblemAreas]
List of objects representing unsimplified face problem areas.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**find_interferences**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *cut_smaller_body: bool = False*) → list[*ansys.geometry.core.tools.problem_areas.InterferenceProblemAreas*]

Find the interference problem areas.

This method finds and returns a list of ids of interference problem areas objects.

Parameters

bodies
[list[Body]] List of bodies that small faces are investigated on.

cut_smaller_body
[bool, optional] Whether to cut the smaller body if an intererference is found. By default, False.

Returns

list[InterfenceProblemAreas]
List of objects representing interference problem areas.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**find_and_fix_short_edges**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *length: ansys.geometry.core.typing.Real = 0.0*, *comprehensive_result: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and fix the short edge problem areas.

This method finds the short edges in the bodies and fixes them.

Parameters

bodies
[list[Body]] List of bodies that short edges are investigated on.

length
[Real, optional] The maximum length of the edges. By default, 0.0.

comprehensive_result
[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns

RepairToolMessage
Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**find_and_fix_extra_edges**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *comprehensive_result: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and fix the extra edge problem areas.

This method finds the extra edges in the bodies and fixes them.

Parameters

bodies
[list[Body]] List of bodies that short edges are investigated on.

length
[Real] The maximum length of the edges.

comprehensive_result
[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns

RepairToolMessage
Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**find_and_fix_split_edges**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *angle: ansys.geometry.core.typing.Real = 0.0*, *length: ansys.geometry.core.typing.Real = 0.0*, *comprehensive_result: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and fix the split edge problem areas.

This method finds the extra edges in the bodies and fixes them.

Parameters

bodies
[list[Body]] List of bodies that split edges are investigated on.

angle
[Real, optional] The maximum angle between edges. By default, 0.0.

length
[Real, optional] The maximum length of the edges. By default, 0.0.

comprehensive_result
[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns

RepairToolMessage
Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**find_and_fix_simplify**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *comprehensive_result: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and simplify the provided geometry.

This method simplifies the provided geometry.

Parameters

bodies
[list[Body]] List of bodies to be simplified.

comprehensive_result
[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns

RepairToolMessage
Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**find_and_fix_stitch_faces**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *max_distance: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real | None = None*, *allow_multiple_bodies: bool = False*, *maintain_components: bool = True*, *check_for_coincidence: bool = False*, *comprehensive_result: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Find and fix the stitch face problem areas.

This method finds the stitchable faces and fixes them.

Parameters

bodies
[list[Body]] List of bodies that stitchable faces are investigated on.

max_distance
[Distance | Quantity | Real, optional] The maximum distance between faces to be stitched. By default, 0.0001.

allow_multiple_bodies
[bool, optional] Whether to allow multiple bodies in the result. By default, False.

maintain_components
[bool, optional] Whether to stitch bodies within the components. By default, True.

check_for_coincidence
[bool, optional] Whether coincidence surfaces are searched. By default, False.

comprehensive_result
[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns

RepairToolMessage
Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

RepairTools.**inspect_geometry**(*bodies: list[ansys.geometry.core.designer.body.Body] = None*) → list[*ansys.geometry.core.tools.check_geometry.InspectResult*]

Return a list of geometry issues organized by body.

This method inspects the geometry and returns a list of the issues grouped by the body where they are found.

Parameters

bodies
[list[Body]] List of bodies to inspect the geometry for. All bodies are inspected if the argument is not given.

Returns

list[IssuesByBody]
List of objects representing geometry issues and the bodies where issues are found.

RepairTools.**repair_geometry**(*bodies: list[ansys.geometry.core.designer.body.Body] = None*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Attempt to repair the geometry for the given bodies.

This method inspects the geometry for the given bodies and attempts to repair them.

Parameters

bodies
[list[Body]] List of bodies where to attempt to repair the geometry. All bodies are repaired if the argument is not given.

Returns

RepairToolMessage
Message containing success of the operation.

Warning

This method is only available starting on Ansys release 25R2.

Description

Provides tools for repairing bodies.

The `unsupported.py` module

Summary

Classes

UnsupportedCommands	Provides unsupported commands for PyAnsys Geometry.

Enums

PersistentIdType	Type of persistent id.

UnsupportedCommands

class ansys.geometry.core.tools.unsupported.**UnsupportedCommands**(*grpc_client: ansys.geometry.core.connection.GrpcClient*, *modeler:* ansys.geometry.core.modeler.Modeler)

Provides unsupported commands for PyAnsys Geometry.

Parameters

grpc_client
[GrpcClient] gRPC client to use for the geometry commands.

modeler
[Modeler] Modeler instance to use for the geometry commands.

Overview

Methods

set_export_id	Set the persistent id for the moniker.
get_body_occurrences_from_import_id	Get all body occurrences whose master has the given import id.
get_face_occurrences_from_import_id	Get all face occurrences whose master has the given import id.
get_edge_occurrences_from_import_id	Get all edge occurrences whose master has the given import id.

Import detail

```
from ansys.geometry.core.tools.unsupported import UnsupportedCommands
```

Method detail

UnsupportedCommands.**set_export_id**(*moniker: str*, *id_type:* PersistentIdType, *value: str*) → None

Set the persistent id for the moniker.

Parameters

moniker
[str] Moniker to set the id for.

id_type
[PersistentIdType] Type of id.

value
[str] Id to set.

Warning

This method is only available starting on Ansys release 25R2.

UnsupportedCommands.**get_body_occurrences_from_import_id**(*import_id: str*, *id_type:* PersistentIdType) → list[*ansys.geometry.core.designer.body.Body*]

Get all body occurrences whose master has the given import id.

Parameters

import_id
[str] Persistent id

id_type
[PersistentIdType] Type of id

Returns

list[Body]
List of body occurrences.

UnsupportedCommands.**get_face_occurrences_from_import_id**(*import_id: str*, *id_type:* PersistentIdType) → list[*ansys.geometry.core.designer.face.Face*]

Get all face occurrences whose master has the given import id.

Parameters

import_id
[str] Persistent id.

id_type
[PersistentIdType] Type of id.

Returns

list[Face]
List of face occurrences.

UnsupportedCommands.**get_edge_occurrences_from_import_id**(*import_id: str*, *id_type:* PersistentIdType) → list[*ansys.geometry.core.designer.edge.Edge*]

Get all edge occurrences whose master has the given import id.

Parameters

import_id
[str] Persistent id.

id_type
[PersistentIdType] Type of id.

Returns

list[Edge]
List of edge occurrences.

PersistentIdType

class ansys.geometry.core.tools.unsupported.**PersistentIdType**(**args*, ***kwds*)

Bases: enum.Enum

Type of persistent id.

Overview

Attributes

PNAME
PRIME_ID

Import detail

```
from ansys.geometry.core.tools.unsupported import PersistentIdType
```

Attribute detail

PersistentIdType.**PNAME = 1**

PersistentIdType.**PRIME_ID = 700**

Description

Unsupported functions for the PyAnsys Geometry library.

Description

PyAnsys Geometry tools subpackage.

The errors.py module

Summary

Exceptions

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*, ** (Keyword-only parameters separator (PEP 3102))*, *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
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.**level = 0**

Logger.**debug**

Logger.**info**

Logger.**warning**

Logger.**error**

Logger.**critical**

Logger.**log**

Method detail

Logger.**log_to_file**(*filename=FILE_NAME*, *level=LOG_LEVEL*)

Add a file handler to the logger.

Parameters

filename
[str, default: "pyansys-geometry.log"] Name of the file to write log messages to.

level
[int, default: 10] Level of logging. The default is `10`, in which case the `logging.DEBUG` level is used.

Examples

Write to the `"pyansys-geometry.log"` file in the current working directory:

```
>>> from ansys.geometry.core import LOG
>>> import os
>>> file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
>>> LOG.log_to_file(file_path)
```

Logger.**log_to_stdout**(*level=LOG_LEVEL*)

Add the standard output handler to the logger.

Parameters

level
[int, default: 10] Level of logging. The default is `10`, in which case the `logging.DEBUG` level is used.

Logger.**setLevel**(*level='DEBUG'*)

Change the log level of the object and the attached handlers.

Logger.**add_child_logger**(*suffix: str*, *level: str | None = None*)

Add a child logger to the main logger.

This logger is more general than an instance logger, which is designed to track the state of Geometry service instances.

If the logging level is in the arguments, a new logger with a reference to the `_global` logger handlers is created instead of a child logger.

Parameters

suffix
[str] Name of the child logger.

level
[str, default: None] Level of logging.

Returns

`logging.Logger`
Logger class.

Logger.**add_instance_logger**(*name: str*, *client_instance: ansys.geometry.core.connection.client.GrpcClient*, *level: int | None = None*) → *PyGeometryCustomAdapter*

Add a logger for a Geometry service instance.

The Geometry service instance logger is a logger with an adapter that adds contextual information such as the Geometry service instance name. This logger is returned, and you can use it to log events as a normal logger. It is stored in the `_instances` field.

Parameters

name
[`str`] Name for the new instance logger.

client_instance
[`GrpcClient`] Geometry service GrpcClient object, which should contain the `get_name` method.

level
[`int`, default: `None`] Level of logging.

Returns

PyGeometryCustomAdapter
Logger adapter customized to add Geometry service information to the logs. You can use this class to log events in the same way you would with the `Logger` class.

`Logger.__getitem__`(*key*)

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

`Logger.add_handling_uncaught_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 = 10**

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

logger.**DEBUG = 10**

logger.**INFO = 20**

logger.**WARN = 30**

logger.**ERROR = 40**

logger.**CRITICAL = 50**

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

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

logger.**DEFAULT_STDOUT_HEADER = Multiline-String**

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

logger.**DEFAULT_FILE_HEADER = Multiline-String**

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

logger.**NEW_SESSION_HEADER = Multiline-String**

```
"""
================================================================================
       NEW SESSION - Uninferable
================================================================================"""
```

logger.**LOG**

logger.**string_to_loglevel**

The `modeler.py` module

Summary

Classes

Modeler	Provides for interacting with an open session of the Geometry service.

Modeler

class ansys.geometry.core.modeler.**Modeler**(*host: str = pygeom_defaults.DEFAULT_HOST*, *port: str | int = pygeom_defaults.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*)

Provides for interacting with an open session of the Geometry service.

Parameters

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

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

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

remote_instance
[ansys.platform.instancemanagement.Instance, default: None] Corresponding remote instance when the Geometry service is launched using PyPIM. This instance is deleted when the GrpcClient.close method is called.

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

product_instance
[ProductInstance, default: None] Corresponding local product instance when the product (Discovery or SpaceClaim) is launched through the launch_modeler_with_geometry_service(), launch_modeler_with_discovery() or the launch_modeler_with_spaceclaim() interface. This instance will be deleted when the GrpcClient.close method is called.

timeout
[Real, default: 120] Time in seconds for trying to achieve the connection.

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

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

Overview

Methods

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.
get_service_logs	Get the service logs.

Properties

client	Modeler instance client.
design	Retrieve the design within the modeler workspace.
designs	Retrieve the design within the modeler workspace.
repair_tools	Access to repair tools.
prepare_tools	Access to prepare tools.
measurement_tools	Access to measurement tools.
geometry_commands	Access to geometry commands.
unsupported	Access to unsupported commands.

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.**design**: *ansys.geometry.core.designer.design.Design*

Retrieve the design within the modeler workspace.

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

Retrieve the design within the modeler workspace.

Notes

This method is deprecated. Use the *design()* property instead.

property Modeler.**repair_tools**: *ansys.geometry.core.tools.repair_tools.RepairTools*

Access to repair tools.

property Modeler.**prepare_tools**: *ansys.geometry.core.tools.prepare_tools.PrepareTools*

Access to prepare tools.

property Modeler.**measurement_tools**:
ansys.geometry.core.tools.measurement_tools.MeasurementTools

Access to measurement tools.

Notes

This property is only available starting on Ansys release 24R2.

property Modeler.**geometry_commands**:
ansys.geometry.core.designer.geometry_commands.GeometryCommands

Access to geometry commands.

property Modeler.**unsupported**: *ansys.geometry.core.tools.unsupported.UnsupportedCommands*

Access to unsupported commands.

Method detail

Modeler.**create_design**(*name: str*) → *ansys.geometry.core.designer.design.Design*

Initialize a new design with the connected client.

Parameters

name
[str] Name for the new design.

Returns

Design
Design object created on the server.

Modeler.**get_active_design**(*sync_with_backend: bool = True*) → *ansys.geometry.core.designer.design.Design*

Get the active design on the modeler object.

Parameters

sync_with_backend
[bool, default: True] Whether to sync the active design with the remote service. If set to False, the active design may be out-of-sync with the remote service. This is useful when the active design is known to be up-to-date.

Returns

Design
Design object already existing on the modeler.

Modeler.**read_existing_design**() → *ansys.geometry.core.designer.design.Design*

Read the existing design on the service with the connected client.

Returns

Design
Design object already existing on the server.

Modeler.**close**(*close_design: bool = True*) → None

Access the client's close method.

Parameters

close_design
[bool, default: True] Whether to close the design before closing the client.

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

Access the client's close method.

Parameters

close_design
[bool, default: True] Whether to close the design before closing the client.

Notes

This method is calling the same method as `close()`.

Modeler.**open_file**(*file_path: str | pathlib.Path*, *upload_to_server: bool = True*, *import_options: ansys.geometry.core.misc.options.ImportOptions = ImportOptions()*) → *ansys.geometry.core.designer.design.Design*

Open a file.

This method imports a design into the service. On Windows and Linux, `.scdocx`, `.dsco`, and reader formats are supported. Please see notes for supported reader formats.

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.

Notes

Format and latest supported version

- AutoCAD 2024
- CATIA V5 2024
- CATIA V6 2024
- Creo Parametric 11
- IGES 5.3
- Inventor 2025
- JT 10.10
- NX 2412
- Rhino 8
- Solid Edge 2025
- SOLIDWORKS 2025
- STEP AP242

`Modeler.__repr__()` → str

Represent the modeler as a string.

`Modeler.run_discovery_script_file`(*file_path: str | pathlib.Path*, *script_args: dict[str, str] | None = None*, *import_design: bool = False*, *api_version: int | str | ansys.geometry.core.connection.backend.ApiVersions = None*) → tuple[dict[str, str], *ansys.geometry.core.designer.design.Design* | None]

Run a Discovery script file.

Note

If arguments are passed to the script, they must be in the form of a dictionary. On the server side, the script will receive the arguments as a dictionary of strings, under the variable name `argsDict`. For example, if the script is called with the arguments `run_discovery_script_file(..., script_args = {"length": "20"}, ...)`, the script will receive the dictionary `argsDict` with the key-value pair `{"length": "20"}`.

Note

If an output is expected from the script, it will be returned as a dictionary of strings. The keys and values of the dictionary are the variables and their values that the script returns. However, it is necessary that the script creates a dictionary called `result` with the variables and their values that are expected to be returned. For example, if the script is expected to return the number of bodies in the design, the script should create a dictionary called `result` with the key-value pair `{"numBodies": numBodies}`, where `numBodies` is the number of bodies in the design.

The implied API version of the script should match the API version of the running Geometry Service. DMS API versions 24.1 and later are supported. DMS is a Windows-based modeling service that has been containerized to ease distribution, execution, and remotability operations.

Parameters

file_path
[`str` | `Path`] Path of the file. The extension of the file must be included.

script_args
[`dict`[`str`, `str`], optional.] Arguments to pass to the script. By default, `None`.

import_design
[bool, optional.] Whether to refresh the current design from the service. When the script is expected to modify the existing design, set this to `True` to retrieve up-to-date design data. When this is set to `False` (default) and the script modifies the current design, the design may be out-of-sync. By default, `False`.

api_version
[`int` | `str` | `ApiVersions`, `optional`] The scripting API version to use. For example, version 24.1 can be passed as an integer 241, a string "241" or using the `ansys.geometry.core.connection.backend.ApiVersions` enum class. By default, `None`. When specified, the service will attempt to run the script with the specified API version. If the API version is not supported, the service will raise an error. If you are using Discovery or SpaceClaim, the product will determine the API version to use, so there is no need to specify this parameter.

Returns

`dict`[`str`, `str`]
Values returned from the script.

`Design`, `optional`
Up-to-date current design. This is only returned if `import_design=True`.

Raises

`GeometryRuntimeError`
If the Discovery script fails to run. Otherwise, assume that the script ran successfully.

Notes

The Ansys Geometry Service only supports scripts that are of the same version as the running service. Any `api_version` input will be ignored.

`Modeler.get_service_logs`(*all_logs: bool = False*, *dump_to_file: bool = False*, *logs_folder: str | pathlib.Path | None = None*) → str | dict[str, str] | pathlib.Path

Get the service logs.

Parameters

all_logs
[bool, default: `False`] Flag indicating whether all logs should be retrieved. By default, only the current logs are retrieved.

dump_to_file
[bool, default: `False`] Flag indicating whether the logs should be dumped to a file. By default, the logs are not dumped to a file.

logs_folder
[`str`, `Path` or `None`, default: `None`] Name of the folder where the logs should be dumped. This parameter is only used if the `dump_to_file` parameter is set to `True`.

Returns

`str`
Service logs as a string. This is returned if the `dump_to_file` parameter is set to `False`.

`dict[str, str]`
Dictionary containing the logs. The keys are the logs names, and the values are the logs as strings. This is returned if the `all_logs` parameter is set to `True` and the `dump_to_file` parameter is set to `False`.

`Path`
Path to the folder containing the logs (if the `all_logs` parameter is set to `True`) or the path to the log file (if only the current logs are retrieved). The `dump_to_file` parameter must be set to `True`.

Notes

This property is only available starting on Ansys release 25R1.

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 = True

Deprecated constant. Use DISABLE_ACTIVE_DESIGN_CHECK instead.

core.**DISABLE_ACTIVE_DESIGN_CHECK**: bool = False

Global constant for disabling the ensure_design_is_active check.

Only set this to false if you are sure you want to disable this check and your objects will always exist on the server side.

core.**DOCUMENTATION_BUILD**: bool

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

core.**USE_TRACKER_TO_UPDATE_DESIGN**: bool = False

Global constant for checking whether to use the tracker to update designs.

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])
```

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```
Vector3D([x, y, z])
Vector2D([x, y])
```

You can perform standard mathematical operations on points and vectors.

Perform some standard operations on vectors.

[2]:
```
vec_1 = Vector3D([1,0,0]) # x-vector
vec_2 = Vector3D([0,1,0]) # y-vector

print("Sum of vectors [1, 0, 0] + [0, 1, 0]:")
print(vec_1 + vec_2) # sum

print("\nDot product of vectors [1, 0, 0] * [0, 1, 0]:")
print(vec_1 * vec_2) # dot

print("\nCross product of vectors [1, 0, 0] % [0, 1, 0]:")
print(vec_1 % vec_2) # cross
```

```
Sum of vectors [1, 0, 0] + [0, 1, 0]:
[1 1 0]

Dot product of vectors [1, 0, 0] * [0, 1, 0]:
0

Cross product of vectors [1, 0, 0] % [0, 1, 0]:
[0 0 1]
```

Create a vector from two points.

[3]:
```
p1 = Point3D([12.4, 532.3, 89])
p2 = Point3D([-5.7, -67.4, 46.6])

vec_3 = Vector3D.from_points(p1, p2)
vec_3
```

[3]:
```
Vector3D([ -18.1, -599.7,  -42.4])
```

Normalize a vector to create a unit vector, which is also known as a *direction*.

[4]:
```
print("Magnitude of vec_3:")
print(vec_3.magnitude)

print("\nNormalized vec_3:")
print(vec_3.normalize())

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

```
Magnitude of vec_3:
601.4694173438911

Normalized vec_3:
[-0.03009297 -0.99705818 -0.07049402]
```

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```
New magnitude:
1.0
```

Use the `UnitVector` class to automatically normalize the input for the unit vector.

[5]:
```
uv = UnitVector3D([1,1,1])
uv
```

[5]:
```
UnitVector3D([0.57735027, 0.57735027, 0.57735027])
```

Perform a few more mathematical operations on vectors.

[6]:
```
v1 = Vector3D([1, 0, 0])
v2 = Vector3D([0, 1, 0])

print("Vectors are perpendicular:")
print(v1.is_perpendicular_to(v2))

print("\nVectors are parallel:")
print(v1.is_parallel_to(v2))

print("\nVectors are opposite:")
print(v1.is_opposite(v2))

print("\nAngle between vectors:")
print(v1.get_angle_between(v2))
print(f"{np.pi / 2} == pi/2")
```

```
Vectors are perpendicular:
True

Vectors are parallel:
False

Vectors are opposite:
False

Angle between vectors:
1.5707963267948966 radian
1.5707963267948966 == pi/2
```

Create planes

Once you begin creating sketches and bodies, `Plane` objects become very important. A plane is defined by these items:

- An origin, which consists of a 3D point
- Two directions (`direction_x` and `direction_y`), which are both `UnitVector3D`objects

If no direction vectors are provided, the plane defaults to the XY plane.

Create two planes.

[7]:
```
plane = Plane(Point3D([0,0,0])) # XY plane
```

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```
print("(1, 2, 0) is in XY plane:")
print(plane.is_point_contained(Point3D([1, 2, 0]))) # True

print("\n(0, 0, 5) is in XY plane:")
print(plane.is_point_contained(Point3D([0, 0, 5]))) # False
```

```
(1, 2, 0) is in XY plane:
True

(0, 0, 5) is in XY plane:
False
```

Perform parametric evaluations

PyAnsys Geometry implements parametric evaluations for some curves and surfaces.

Evaluate a sphere.

[8]:
```
from ansys.geometry.core.shapes import Sphere, SphereEvaluation
from ansys.geometry.core.math import Point3D
from ansys.geometry.core.misc import Distance

sphere = Sphere(Point3D([0,0,0]), Distance(1)) # radius = 1

eval = sphere.project_point(Point3D([1,1,1]))

print("U Parameter:")
print(eval.parameter.u)

print("\nV Parameter:")
print(eval.parameter.v)
```

```
U Parameter:
0.7853981633974483

V Parameter:
0.6154797086703873
```

[9]:
```
print("Point on the sphere:")
eval.position
```

```
Point on the sphere:
```

[9]:
```
Point3D([0.57735027, 0.57735027, 0.57735027])
```

[10]:
```
print("Normal to the surface of the sphere at the evaluation position:")
eval.normal
```

```
Normal to the surface of the sphere at the evaluation position:
```

[10]:
```
UnitVector3D([0.57735027, 0.57735027, 0.57735027])
```

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8.1.2 PyAnsys Geometry 101: Units

To handle units inside the source code, PyAnsys Geometry uses Pint, a third-party open source software that other PyAnsys libraries also use.

The following code examples show how to operate with units inside the PyAnsys Geometry codebase and create objects with different units.

Import units handler

The following line of code imports the units handler: `pint.util.UnitRegistry`. For more information on the `UnitRegistry` class in the `pint` API, see Most important classes in the Pint documentation.

[1]:
```
from ansys.geometry.core.misc import UNITS
```

Create and work with `Quantity` objects

With the `UnitRegistry` object called `UNITS`, you can create `Quantity` objects. A `Quantity` object is simply a container class with two core elements:

- A number
- A unit

`Quantity` objects have convenience methods, including those for transforming to different units and comparing magnitudes, values, and units. For more information on the `Quantity` class in the `pint` API, see Most important classes in the Pint documentation. You can also step through this Pint tutorial.

[2]:
```
from pint import Quantity

a = Quantity(10, UNITS.mm)

print(f"Object a is a pint.Quantity: {a}")

print("Request its magnitude in different ways (accessor methods):")
print(f"Magnitude: {a.m}.")
print(f"Also magnitude: {a.magnitude}.")

print("Request its units in different ways (accessor methods):")
print(f"Units: {a.u}.")
print(f"Also units: {a.units}.")
```

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```
# Quantities can be compared between different units
# You can also build Quantity objects as follows:
a2 = 10 * UNITS.mm
print(f"Compare quantities built differently: {a == a2}")

# Quantities can be compared between different units
a2_diff_units = 1 * UNITS.cm
print(f"Compare quantities with different units: {a == a2_diff_units}")
```

```
Object a is a pint.Quantity: 10 millimeter
Request its magnitude in different ways (accessor methods):
Magnitude: 10.
Also magnitude: 10.
Request its units in different ways (accessor methods):
Units: millimeter.
Also units: millimeter.
Compare quantities built differently: True
Compare quantities with different units: True
```

PyAnsys Geometry objects work by returning `Quantity` objects whenever the property requested has a physical meaning.

Return `Quantity` objects for `Point3D` objects.

[3]:
```
from ansys.geometry.core.math import Point3D

point_a = Point3D([1,2,4])
print("========================= Point3D([1,2,4]) =========================")
print(f"Point3D is a numpy.ndarray in SI units: {point_a}.")
print(f"However, request each of the coordinates individually...\n")
print(f"X Coordinate: {point_a.x}")
print(f"Y Coordinate: {point_a.y}")
print(f"Z Coordinate: {point_a.z}\n")

# Now, store the information with different units...
point_a_km = Point3D([1,2,4], unit=UNITS.km)
print("================= Point3D([1,2,4], unit=UNITS.km) ==================")
print(f"Point3D is a numpy.ndarray in SI units: {point_a_km}.")
print(f"However, request each of the coordinates individually...\n")
print(f"X Coordinate: {point_a_km.x}")
print(f"Y Coordinate: {point_a_km.y}")
print(f"Z Coordinate: {point_a_km.z}\n")

# These points, although they are in different units, can be added together.
res = point_a + point_a_km

print("==================== res = point_a + point_a_km ====================")
print(f"numpy.ndarray: {res}")
print(f"X Coordinate: {res.x}")
print(f"Y Coordinate: {res.y}")
print(f"Z Coordinate: {res.z}")
```

```
========================= Point3D([1,2,4]) =========================
Point3D is a numpy.ndarray in SI units: [1. 2. 4.].
```

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```
However, request each of the coordinates individually...

X Coordinate: 1 meter
Y Coordinate: 2 meter
Z Coordinate: 4 meter

================= Point3D([1,2,4], unit=UNITS.km) =================
Point3D is a numpy.ndarray in SI units: [1000. 2000. 4000.].
However, request each of the coordinates individually...

X Coordinate: 1 kilometer
Y Coordinate: 2 kilometer
Z Coordinate: 4 kilometer

=================== res = point_a + point_a_km ====================
numpy.ndarray: [1001. 2002. 4004.]
X Coordinate: 1001.0 meter
Y Coordinate: 2002.0 meter
Z Coordinate: 4004.0 meter
```

Use default units

PyAnsys Geometry implements the concept of *default units*.

[4]:
```
from ansys.geometry.core.misc import DEFAULT_UNITS

print("=== Default unit length ===")
print(DEFAULT_UNITS.LENGTH)

print("=== Default unit angle ===")
print(DEFAULT_UNITS.ANGLE)
```

```
=== Default unit length ===
meter
=== Default unit angle ===
radian
```

It is important to differentiate between *client-side* default units and *server-side* default units. You are able to control both of them.

Print the default server unit length.

[5]:
```
print("=== Default server unit length ===")
print(DEFAULT_UNITS.SERVER_LENGTH)
```

```
=== Default server unit length ===
meter
```

Use default units.

[6]:
```
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import DEFAULT_UNITS

DEFAULT_UNITS.LENGTH = UNITS.mm
```

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```
point_2d_default_units = Point2D([3, 4])
print("This is a Point2D with default units")
print(f"X Coordinate: {point_2d_default_units.x}")
print(f"Y Coordinate: {point_2d_default_units.y}")
print(f"numpy.ndarray value: {point_2d_default_units}")

# Revert back to original default units
DEFAULT_UNITS.LENGTH = UNITS.m
```

```
This is a Point2D with default units
X Coordinate: 3 millimeter
Y Coordinate: 4 millimeter
numpy.ndarray value: [0.003 0.004]
```

PyAnsys Geometry has certain auxiliary classes implemented that provide proper unit checking when assigning values. Although they are basically intended for internal use of the library, you can define them for use.

[7]:
```
from ansys.geometry.core.misc import Angle, Distance
```

Start with `Distance`. The main difference between a `Quantity` object (that is, `from pint import Quantity`) and a `Distance` is that there is an active check on the units passed (in case they are not the default ones). Here are some examples.

[8]:
```
radius = Distance(4)
print(f"The radius is {radius.value}.")

# Reassign the value of the distance
radius.value = 7 * UNITS.cm
print(f"After reassignment, the radius is {radius.value}.")


# Change the units if desired
radius.unit = UNITS.cm
print(f"After changing its units, the radius is {radius.value}.")
```

```
The radius is 4 meter.
After reassignment, the radius is 0.07 meter.
After changing its units, the radius is 7.000000000000001 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]))
```

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```
sketch = Sketch(plane)

sketch.box(Point2D([0,0]), 1, 1)

sketch.plot()
```

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

Combine concepts to create powerful sketches

Combine these simple concepts to create powerful sketches.

[5]:
```
# Complex Fluent API Sketch - PCB

sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↪"LowerEdge")
      .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↪mm), tag="SupportedCorner")
      .segment_to_point(Point2D([41.5, 5], unit=UNITS.mm))
      .arc_to_point(Point2D([43, 6.5], unit=UNITS.mm), Point2D([43, 5], unit=UNITS.mm),␣
↪True)
      .segment_to_point(Point2D([55, 6.5], unit=UNITS.mm))
      .arc_to_point(Point2D([56.5, 8], unit=UNITS.mm), Point2D([55, 8], unit=UNITS.mm))
      .segment_to_point(Point2D([56.5, 35], unit=UNITS.mm))
      .arc_to_point(Point2D([55, 36.5], unit=UNITS.mm), Point2D([55, 35], unit=UNITS.mm))
      .segment_to_point(Point2D([0, 36.5], unit=UNITS.mm))
      .segment_to_point(Point2D([0, 0], unit=UNITS.mm))
      .circle(Point2D([4, 4], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor1")
      .circle(Point2D([51, 34.5], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor2")
)

sketch.plot()
```

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

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

```
Ansys Geometry Modeler (0x7faf37f092b0)

Ansys Geometry Modeler Client (0x7faf37f0ba10)
  Target:     localhost:700
  Connection: Healthy
```

You can also launch your own services and connect to them. For information on connecting to an existing service, see the Modeler API documentation.

Here is how the class architecture is implemented:

- `Modeler`: Handler object for the active service session. This object allows you to connect to an existing service by passing in a host and a port. It also allows you to create `Design` objects, which is where the modeling takes place. For more information, see the Modeler API documentation.
- `Design`: Root object of your assembly (tree). While a `Design` object is also a `Component` object, it has enhanced capabilities, including creating named selections, adding materials, and handling beam profiles. For more information, see the Design API documentation.
- `Component`: One of the main objects for modeling purposes. `Component` objects allow you to create bodies, subcomponents, beams, design points, planar surfaces, and more. For more information, see the Component API documentation.

The following code examples show how you use these objects. More capabilities of these objects are shown in the specific example sections for sketching and modeling.

Create and plot a sketch

Create a Sketch object and plot it.

[2]:
```
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS, Distance

outer_hole_radius = Distance(0.5, UNITS.m)

sketch = Sketch()
(
    sketch.segment(start=Point2D([-4, 5], unit=UNITS.m), end=Point2D([4, 5], unit=UNITS.
↪m))
    .segment_to_point(end=Point2D([4, -5], unit=UNITS.m))
    .segment_to_point(end=Point2D([-4, -5], unit=UNITS.m))
    .segment_to_point(end=Point2D([-4, 5], unit=UNITS.m))
    .box(
        center=Point2D([0, 0], unit=UNITS.m),
        width=Distance(3, UNITS.m),
        height=Distance(3, UNITS.m),
    )
    .circle(center=Point2D([3, 4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([-3, -4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([-3, 4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([3, -4], unit=UNITS.m), radius=outer_hole_radius)
)

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

```
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 54.28672587712814 meter ** 3.
```

Other operations that can be performed include adding a midsurface offset and thickness (only for planar bodies), imprinting curves, assigning materials, copying, and translating.

Copy the body on a new subcomponent and translate it.

[5]:
```
from ansys.geometry.core.math import UNITVECTOR3D_X

# Create a component
comp = design.add_component("Component")

# Copy the body that belongs to this new component
body_copy = body.copy(parent=comp, name="Design_Component_Body")

# Displace this new body by a certain distance (10m) in a certain direction (X-axis)
body_copy.translate(direction=UNITVECTOR3D_X, distance=Distance(10, unit=UNITS.m))

# Plot the result of the entire design
design.plot()
```

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

Create and assign materials to the bodies that were created.

[6]:
```
from pint import Quantity

from ansys.geometry.core.materials import Material, MaterialProperty,␣
↪MaterialPropertyType

# Define some general properties for the material.
density = Quantity(125, 10 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
poisson_ratio = Quantity(0.33, UNITS.dimensionless)
tensile_strength = Quantity(45)  # WARNING: If no units are defined,
#it is assumed that the magnitude is in the units expected by the server.

# Once your material properties are defined, you can easily create a material.
material = Material(
    "steel",
    density,
```

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```
    [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.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 (0x7efffd8af380)

Ansys Geometry Modeler Client (0x7efffd7c8050)
  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)
```

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

Clip objects

You can clip any object represented in the plotter by defining a `Plane` object that intersects the target object.

[11]:
```
from ansys.geometry.core.math import Plane, Point3D
pl = GeometryPlotter()

# Define PyAnsys Geometry box
box2 = Sketch()
box2.box(Point2D([-10, 20], UNITS.m), Quantity(10, UNITS.m), Quantity(10, UNITS.m))
box_body2 = design.extrude_sketch("JustABox", box2, Quantity(10, UNITS.m))

# Define plane to clip the box
origin = Point3D([-10., 20., 5.], UNITS.m)
plane = Plane(origin=origin, direction_x=[1, 1, 1], direction_y=[-1, 0, 1])

# Add the object with the clipping plane
pl.plot(box_body2, clipping_plane=plane)
pl.show()
```

```
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↪http-equiv=&quot;Content-...
```

Select objects interactively

PyAnsys Geometry's plotter supports interactive object selection within the scene. This enables you to pick objects for subsequent script manipulation.

[12]:
```
plotter = GeometryPlotter(allow_picking=True)

# Plotter returns picked bodies
picked_list = plotter.show(plot_list)
print(picked_list)
```

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

```
None
```

It is also possible to enable picking directly for a specific `design` or `component` object alone. In the following cell, picking is enabled for the `design` object.

[13]:
```
picked_list = design.plot(allow_picking=True)
print(picked_list)
```

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

```
None
```

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 you can create a new design and plot a prism and a cylinder in different colors.

[14]:
```
design = modeler.create_design("MultiColors")

# Create a sketch of a box
sketch_box = Sketch().box(Point2D([0, 0], unit=UNITS.m), width=30 * UNITS.m, height=40 *␣
↪UNITS.m)

# Create a sketch of a circle (overlapping the box slightly)
sketch_circle = Sketch().circle(Point2D([20, 0], unit=UNITS.m), radius=3 * UNITS.m)

# Extrude both sketches to get a prism and a cylinder
design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

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

```
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.

[15]:
```
modeler.close()
```

Download this example

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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(
```

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```
    origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
)

# Instantiate a new sketch object from previous plane
sketch = Sketch(plane)
```

Draw shapes

To draw different shapes in the sketch, you use `draw` methods.

Draw a circle

You draw a circle in a sketch by specifying the center and radius.

[4]:
```
sketch.circle(Point2D([2, 1]), radius=30 * u.cm, tag="Circle")
sketch.select("Circle")
sketch.plot_selection()
```

```
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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Download this example

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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]:
```
modeler = launch_modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x7f4cb29cc440)

Ansys Geometry Modeler Client (0x7f4cb29cc590)
  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]:
```
design = modeler.create_design("ExtrudedBoardProfile")

layers = []
layer_thickness = Quantity(0.20, UNITS.mm)
for layer_index in range(10):
    layers.append(design.extrude_sketch(f"BoardLayer_{layer_index}", sketch, layer_
↪thickness))
    sketch.translate_sketch_plane_by_distance(UNITVECTOR3D_Z, layer_thickness)

board_named_selection = design.create_named_selection("FullBoard", bodies=layers)
design.plot()
```

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

Close the modeler

Close the modeler to release the resources.

[5]:
```
modeler.close()
```

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8.2.3 Sketching: Parametric sketching for gears

This example shows how to use gear sketching shapes from PyAnsys Geometry.

Perform required imports and pre-sketching operations

Perform required imports and instantiate the `Modeler` instance and the basic elements that define a sketch.

[1]:
```
from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point2D, Point3D
from ansys.geometry.core.misc import UNITS, Distance
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.plotting import GeometryPlotter

# Start a modeler session
modeler = launch_modeler()
print(modeler)

# Define the origin point of the plane
origin = Point3D([1, 1, 1])

# Create a plane containing the previous point with desired fundamental directions
plane = Plane(
    origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
)
```

```
Ansys Geometry Modeler (0x7efd9f76b380)

Ansys Geometry Modeler Client (0x7efd9f684050)
  Target:     localhost:700
  Connection: Healthy
```

Sketch a dummy gear

`DummyGear` sketches are simple gears that have straight teeth. While they do not ensure actual physical functionality, they might be useful for some simple playground tests.

Instantiate a new `Sketch` object and then define and plot a dummy gear.

[2]:
```
# Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

# Define dummy gear
#
origin = Point2D([0, 1], unit=UNITS.meter)
outer_radius = Distance(4, unit=UNITS.meter)
inner_radius = Distance(3.8, unit=UNITS.meter)
n_teeth = 30
sketch.dummy_gear(origin, outer_radius, inner_radius, n_teeth)

# Plot dummy gear
sketch.plot()
```

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

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(continued from previous page)

```
# Extrude sketch
dummy_gear = design.extrude_sketch("SpurGear", sketch, Distance(200, UNITS.mm))

# Plot design
design.plot()
```

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

Close the modeler

[6]:
```
modeler.close()
```

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8.3 Modeling examples

These examples demonstrate service-based modeling operations.

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8.3.1 Modeling: Single body with material assignment

In PyAnsys Geometry, a *body* represents solids or surfaces organized within the `Design` assembly. The current state of sketch, which is a client-side execution, can be used for the operations of the geometric design assembly.

The Geometry service provides data structures to create individual materials and their properties. These data structures are exposed through PyAnsys Geometry.

This example shows how to create a single body from a sketch by requesting its extrusion. It then shows how to assign a material to this body.

Perform required imports

Perform the required imports.

[1]:
```
from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.materials import Material, MaterialProperty,␣
↪MaterialPropertyType
from ansys.geometry.core.math import UNITVECTOR3D_Z, Frame, Plane, Point2D, Point3D,␣
```

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```
↪UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Create sketch

Create a `Sketch` instance and insert a circle with a radius of 10 millimeters in the default plane.

[2]:
```
sketch = Sketch()
sketch.circle(Point2D([10, 10], UNITS.mm), Quantity(10, UNITS.mm))
```

[2]:
```
<ansys.geometry.core.sketch.sketch.Sketch at 0x7f40a4da41a0>
```

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 (0x7f40a4da63c0)

Ansys Geometry Modeler Client (0x7f40a4c7cc20)
  Target:     localhost:700
  Connection: Healthy
```

Add materials to design

Add materials and their properties to the design. Material properties can be added when creating the `Material` object or after its creation. This code adds material properties after creating the `Material` object.

[4]:
```
density = Quantity(125, 10 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
poisson_ratio = Quantity(0.33, UNITS.dimensionless)
tensile_strength = Quantity(45)
material = Material(
    "steel",
    density,
    [MaterialProperty(MaterialPropertyType.POISSON_RATIO, "PoissonRatio", poisson_
↪ratio)],
)
material.add_property(MaterialPropertyType.TENSILE_STRENGTH, "TensileProp", Quantity(45))
design.add_material(material)
```

Extrude sketch to create body

Extrude the sketch to create the body and then assign a material to it.

[5]:
```
# Extrude the sketch to create the body
body = design.extrude_sketch("SingleBody", sketch, Quantity(10, UNITS.mm))
```

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```
# Assign a material to the body
body.assign_material(material)

body.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&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 *Sketch* user guide.

In this example, a box is located in the center of the plate, with the default origin of a sketch plane (origin at `(0, 0, 0)`). Four holes of equal radius are sketched at the corners of the plate. The plate is then extruded, leading to the generation of the requested body. The projection is at the center of the face. The default projection depth is through the entire part.

Perform required imports

Perform the required imports.

[1]:
```
import numpy as np
from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point3D, Point2D, UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile

The sketch profile for the proposed design requires four segments that constitute the outer limits of the design, a box on the center, and a circle at its four corners.

You can use a single `sketch` instance for multiple design operations, including extruding a body, projecting a profile, and imprinting curves.

Define the sketch 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 0x7f33fa4201a0>
```

Extrude sketch to create design

Establish a server connection and use the single sketch profile to extrude the base component at the Z axis. Create a named selection from the resulting list of bodies. In only three server calls, the design extrudes the four segments with the desired thickness.

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

design = modeler.create_design("ExtrudedPlate")

body = design.extrude_sketch(f"PlateLayer", sketch, Quantity(2, UNITS.m))

board_named_selection = design.create_named_selection("Plate", bodies=[body])
design.plot()
```

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

Add component with a planar surface

After creating a plate as a base component, you might want to add a component with a planar surface to it.

Create a `sketch` instance and then create a surface in the design with this sketch. For the sketch, it creates an ellipse, keeping the origin of the plane as its center.

[4]:
```
# Add components to the design
planar_component = design.add_component("PlanarComponent")

# Initiate ``Sketch`` to create the planar surface.
planar_sketch = Sketch()
planar_sketch.ellipse(
        Point2D([0, 0], UNITS.m), Quantity(1, UNITS.m), Quantity(0.5, UNITS.m)
```

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```
    )

planar_body = planar_component.create_surface("PlanarComponentSurface", planar_sketch)

comp_str = repr(planar_component)
design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&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]:
```
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: Extruded plate with cut operations

As seen in the *Rectangular plate with multiple bodies* example, you can create a complex sketch with holes and extrude it to create a body. However, you can also perform cut operations on the extruded body to achieve similar results.

Perform required imports

Perform the required imports.

[1]:
```
from pint import Quantity

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

Define sketch profile without holes

Create a sketch profile for the proposed design. The sketch is the same as the *Rectangular plate with multiple bodies* example, but without the holes.

These holes are created by performing cut operations on the extruded body in the next steps.

[2]:
```
sketch = Sketch()
(sketch.segment(Point2D([-4, 5], unit=UNITS.m), Point2D([4, 5], unit=UNITS.m))
    .segment_to_point(Point2D([4, -5], unit=UNITS.m))
    .segment_to_point(Point2D([-4, -5], unit=UNITS.m))
    .segment_to_point(Point2D([-4, 5], unit=UNITS.m))
    .box(Point2D([0,0], unit=UNITS.m), Quantity(3, UNITS.m), Quantity(3, UNITS.m))
)

modeler = launch_modeler()
design = modeler.create_design("ExtrudedPlateNoHoles")
body = design.extrude_sketch(f"PlateLayer", sketch, Quantity(2, UNITS.m))

design.plot()
```

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

Define sketch profile for holes

Create a sketch profile for the holes in the proposed design. The holes are created by sketching circles at the four corners of the plate. First create a reference sketch for all the circles. This sketch is translated to the four corners of the plate.

[3]:
```
sketch_hole = Sketch()
sketch_hole.circle(Point2D([0, 0], unit=UNITS.m), Quantity(0.5, UNITS.m))

hole_centers = [
    Plane(Point3D([3, 4, 0], unit=UNITS.m)),
    Plane(Point3D([-3, 4, 0], unit=UNITS.m)),
    Plane(Point3D([-3, -4, 0], unit=UNITS.m)),
    Plane(Point3D([3, -4, 0], unit=UNITS.m)),
]
```

Perform cut operations on the extruded body

Perform cut operations on the extruded body to create holes at the four corners of the plate.

[4]:
```
for center in hole_centers:
    sketch_hole.plane = center
    design.extrude_sketch(
        name= f"H_{center.origin.x}_{center.origin.y}",
        sketch=sketch_hole,
        distance=Quantity(2, UNITS.m),
        cut=True,
    )

design.plot()
```

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

Close the modeler

Close the modeler to free up resources and release the connection.

[5]:
```
modeler.close()
```

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8.3.4 Modeling: Tessellation of two bodies

This example shows how to create two stacked bodies and return the tessellation as two merged bodies.

Perform required imports

Perform the required imports.

[1]:
```
from pint import Quantity

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

Create design

Create the basic sketches to be tessellated and extrude the sketch in the required plane. For more information on creating a component and extruding a sketch in the design, see the *Rectangular plate with multiple bodies* example.

Here is a typical situation in which two bodies, with different sketch planes, merge each body into a single dataset. This effectively combines all the faces of each individual body into a single dataset without separating faces.

[2]:
```
modeler = launch_modeler()

sketch_1 = Sketch()
box = sketch_1.box(
    Point2D([10, 10], unit=UNITS.m), width=Quantity(10, UNITS.m), height=Quantity(5,␣
↪UNITS.m)
)
circle = sketch_1.circle(
    Point2D([0, 0], unit=UNITS.m), radius=Quantity(25, UNITS.m)
)

design = modeler.create_design("TessellationDesign")
comp = design.add_component("TessellationComponent")
body = comp.extrude_sketch("Body", sketch=sketch_1, distance=10 * UNITS.m)

# Create the second body in a plane with a different origin
sketch_2 = Sketch(Plane([0, 0, 10]))
box = sketch_2.box(Point2D(
    [10, 10], unit=UNITS.m), width=Quantity(10, UNITS.m), height=Quantity(5, UNITS.m)
)
circle = sketch_2.circle(
    Point2D([0, 10], unit=UNITS.m), radius=Quantity(25, UNITS.m)
)

body = comp.extrude_sketch("Body", sketch=sketch_2, distance=10 * UNITS.m)
```

Tessellate component as two merged bodies

Tessellate the component and merge each body into a single dataset. This effectively combines all the faces of each individual body into a single dataset without separating faces.

[3]:
```
dataset = comp.tessellate()
dataset
```

[3]:
```
PolyData (0x7fca303044c0)
  N Cells:    3280
  N Points:   3300
  N Strips:   0
  X Bounds:   -2.500e+01, 2.500e+01
  Y Bounds:   -2.500e+01, 3.500e+01
  Z Bounds:   0.000e+00, 2.000e+01
  N Arrays:   0
```

Single body tessellation is possible. In that case, users can request the body-level tessellation method to tessellate the body and merge all the faces into a single dataset.

[4]:
```
dataset = comp.bodies[0].tessellate()
dataset
```

[4]:
```
MultiBlock (0x7fca30293700)
  N Blocks:   7
  X Bounds:   -2.500e+01, 2.500e+01
  Y Bounds:   -2.500e+01, 2.500e+01
  Z Bounds:   0.000e+00, 1.000e+01
```

Plot design

Plot the design.

[5]:
```
design.plot()
```

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

Close the modeler

Close the modeler.

[6]:
```
modeler.close()
```

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8.3.5 Modeling: Design organization

The `Design` instance creates a design project within the remote Geometry service to complete all CAD modeling against.

You can organize all solid and surface bodies in each design within a customizable component hierarchy. A component is simply an organization mechanism.

The top-level design node and each child component node can have one or more bodies assigned and one or more components assigned.

The API requires each component of the design hierarchy to be given a user-defined name.

There are several design operations that result in a body being created within a design. Executing each of these methods against a specific component instance explicitly specifies the node of the design tree to place the new body under.

Perform required imports

Perform the required imports.

[1]:
```
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, Point2D
from ansys.geometry.core.misc import UNITS, Distance
from ansys.geometry.core.sketch import Sketch
```

Organize design

Extrude two sketches to create bodies. Assign the cylinder to the top-level design component. Assign the slot to the component nested one level beneath the top-level design component.

[2]:
```
modeler = launch_modeler()

design = modeler.create_design("DesignHierarchyExample")

circle_sketch = Sketch()
circle_sketch.circle(Point2D([10, 10], UNITS.mm), Distance(10, UNITS.mm))

cylinder_body = design.extrude_sketch("10mmCylinder", circle_sketch, Distance(10, UNITS.
↪mm))

slot_sketch = Sketch()
slot_sketch.slot(Point2D([40, 10], UNITS.mm), Distance(20, UNITS.mm), Distance(10, UNITS.
↪mm))

nested_component = design.add_component("NestedComponent")
slot_body = nested_component.extrude_sketch("SlotExtrusion", slot_sketch, Distance(20,␣
↪UNITS.mm))

design.plot()
```

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

Create nested component

Create a component that is nested under the previously created component and then create another cylinder from the previously used sketch.

[3]:
```
double_nested_component = nested_component.add_component("DoubleNestedComponent")

circle_surface_body = double_nested_component.create_surface("CircularSurfaceBody",␣
↪circle_sketch)
circle_surface_body.translate(UNITVECTOR3D_X, Distance(-35, UNITS.mm))

design.plot()
```

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

Use surfaces from body to create additional bodies

You can use surfaces from any body across the entire design as references for creating additional bodies.

Extrude a cylinder from the surface body assigned to the child component.

[4]:
```
cylinder_from_face = nested_component.extrude_face("CylinderFromFace", circle_surface_
↪body.faces[0], Distance(30, UNITS.mm))
cylinder_from_face.translate(UNITVECTOR3D_X, Distance(-25, UNITS.mm))

design.plot()
```

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

Close the modeler

Close the modeler to release the resources.

[5]:
```
modeler.close()
```

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8.3.6 Modeling: Boolean operations

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

Perform required imports

Perform the required imports.

[1]:
```
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.designer import Body
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import GeometryPlotter
from ansys.geometry.core.sketch import Sketch
```

Launch local modeler

Launch the local modeler. If you are not familiar with how to launch the local modeler, see the "Launch a modeling service" section in the *PyAnsys Geometry 101: Modeling* example.

[2]:
```
modeler = launch_modeler()
print(modeler)
```

```
Ansys Geometry Modeler (0x7f957ba442f0)

Ansys Geometry Modeler Client (0x7f957ba44440)
  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 0x7f957b9d5350
  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 0x7f9568768bb0
  Name                 : Prism
  Exists               : True
  Parent component     : union_design
  MasterBody           : 0:22
  Surface body         : False
  Color                : #D6F7D1
]
```

Perform a subtraction operation

To perform a subtraction operation on the bodies, first set up the bodies.

[12]:
```
# Create a design
design = modeler.create_design("subtraction_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

Perform the subtraction and plot the results.

[13]:
```
prism.subtract(cylin)
_ = GeometryPlotter().show(design.bodies)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta␣
↪http-equiv=&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 0x7f95688a0520
  Name                 : Prism
  Exists               : True
  Parent component     : subtraction_design
  MasterBody           : 0:22
  Surface body         : False
  Color                : #D6F7D1
]
```

If you perform this action inverting the order of the bodies (that is, `cylin.subtract(prism)`, you can see the difference in the resulting shape of the body.

[15]:
```
# Create a design
design = modeler.create_design("subtraction_design_inverted")
```

(continues on next page)

(continued from previous page)

```
# 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 0x7f95680ab0d0
  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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Download this example

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8.3.7 Modeling: Scale, map and mirror bodies

The purpose of this notebook is to demonstrate the `map()` and `scale()` functions and their usage for transforming bodies.

[1]:
```
# Imports
import numpy as np

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
  Frame,
  Plane,
  Point2D,
  Point3D,
  UNITVECTOR3D_X,
  UNITVECTOR3D_Y,
  UNITVECTOR3D_Z,
)
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

[2]:
```
# Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
Ansys Geometry Modeler (0x7f22ee4b52b0)

Ansys Geometry Modeler Client (0x7f22ee4b7a10)
  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 is changed.

```
body.scale(value)
```

Example: Making a cube

The following code snippets show how to change the size of a cube using the `scale()` function in `Body` objects. The process involves initializing a sketch design for the cube, defining the shape parameters, and then performing a rescaling operation to generate the new shape.

Initialize the cube sketch design

A new design sketch named “cube” is created.

[3]:
```
design = m.create_design("cube")
```

Define cube parameters

`side_length` is set to 10 units, representing the side length of the cube.

[4]:
```
# Cube parameters
side_length = 10
```

Create the profile cube

A square box is created centered on the origin using `side_length` as the side length of the square.

[5]:
```
# Square with side length 10
box_sketch = Sketch().box(Point2D([0, 0]), side_length, side_length)

box_sketch.plot()
```

```
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()
(
```

(continues on next page)

(continued from previous page)

```
   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()
```

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

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

Mirror the triangular body

First, make a copy of the triangular body. Then, using `mirror()`, you can mirror the copied body over the ZY plane.

[17]:
```
# Copy triangular body
mirrored_triangle = triangle.copy(triangle.parent_component, "mirrored_triangle")
# Mirror the copied body over the ZY plane (specified by the (0, 1, 0) and
# (0, 0, 1) unit vectors)

mirrored_triangle.mirror(Plane(direction_x=UNITVECTOR3D_Y, direction_y=UNITVECTOR3D_Z))

design.plot()
```

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

Closing the modeler

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

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8.3.8 Modeling: Sweep chain and sweep sketch

This example shows how use the `sweep_sketch()` and `sweep_chain()` functions to create more complex extrusion profiles. You use the `sweep_sketch()` function with a closed sketch profile and the `sweep_chain()` function for an open profile.

[1]:
```
# Imports
import numpy as np

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
  Plane,
  Point2D,
  Point3D,
  UNITVECTOR3D_X,
```

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```
  UNITVECTOR3D_Y,
  UNITVECTOR3D_Z,
)
from ansys.geometry.core.shapes import Circle, Ellipse, Interval
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

[2]:
```
# Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
Ansys Geometry Modeler (0x7fa4024412b0)

Ansys Geometry Modeler Client (0x7fa402443a10)
  Target:     localhost:700
  Connection: Healthy
```

Example: Creating a donut

The following code snippets show how to use the `sweep_sketch()` function to create a 3D donut shape in the `Design` object. The process involves initializing a sketch design for the donut, defining two circles for the profile and path, and then performing a sweep operation to generate the donut shape.

Initialize the donut sketch design

A new design sketch named “donut” is created.

[3]:
```
# Initialize the donut sketch design
design_sketch = m.create_design("donut")
```

Define circle parameters

`path_radius` is set to 5 units, representing the radius of the circular path that the profile circle sweeps along. `profile_radius` is set to 2 units, representing the radius of the profile circle that sweeps along the path to create the donut body.

[4]:
```
# Donut parameters
path_radius = 5
profile_radius = 2
```

Create the profile circle

A circle is created on the XZ-plane centered at the coordinates (5, 0, 0) with a radius defined by `profile_radius`. This circle serves as the profile or cross-sectional shape of the donut.

[5]:
```
# Create the circular profile on the XZ plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(direction_x=UNITVECTOR3D_X, direction_y=UNITVECTOR3D_Z)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([path_radius, 0]), profile_radius)
```

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(continued from previous page)

```
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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Download this example

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8.3.9 Modeling: Revolving a sketch

This example shows how to use the `revolve_sketch()` method to revolve a sketch around an axis to create a 3D body. You can also specify the angle of revolution to create a partial body.

[1]:
```
# Imports
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
  Plane,
  Point2D,
  Point3D,
  UNITVECTOR3D_X,
  UNITVECTOR3D_Z,
)
from ansys.geometry.core.misc import UNITS, Angle
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

[2]:
```
# Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
Ansys Geometry Modeler (0x7fc8295692b0)

Ansys Geometry Modeler Client (0x7fc82956ba10)
  Target:     localhost:700
  Connection: Healthy
```

Example: Creating a quarter of a donut

The following code snippets show how to use the `revolve_sketch()` function to create a quarter of a 3D donut. The process involves defining a quarter of a circle as a profile and then revolving it around the Z-axis to create a 3D body.

Initialize the sketch design

Create a design sketch named `quarter-donut`.

[3]:
```
# Initialize the donut sketch design
design = m.create_design("quarter-donut")
```

Define circle parameters

Set `path_radius`, which represents the radius of the circular path that the profile circle sweeps along, to 5 units. Set `profile_radius`, which represents the radius of the profile circle that sweeps along the path to create the donut body, to 2 units.

[4]:
```
# Donut parameters
path_radius = 5
profile_radius = 2
```

Create the profile circle

Create a circle on the XZ plane centered at the coordinates `(5, 0, 0)` and use`profile_radius` to define the radius. This circle serves as the profile or cross-sectional shape of the donut.

[5]:
```
# Create the circular profile on the XZ-plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(
    origin=Point3D([path_radius, 0, 0]),
    direction_x=UNITVECTOR3D_X,
    direction_y=UNITVECTOR3D_Z,
)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([0, 0]), profile_radius)

profile.plot()
```

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

Perform the revolve operation

Revolve the profile circle around the Z axis to create a quarter of a donut body. Set the angle of revolution to 90 degrees in the default direction, which is counterclockwise.

[6]:
```
# Revolve the profile around the Z axis and center in the absolute origin
# for an angle of 90 degrees
design.revolve_sketch(
    "quarter-donut-body",
    sketch=profile,
    axis=UNITVECTOR3D_Z,
    angle=Angle(90, unit=UNITS.degrees),
    rotation_origin=Point3D([0, 0, 0]),
)

design.plot()
```

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

Perform a revolve operation with a negative angle of revolution

You can use a negative angle of revolution to create a quarter of a donut in the opposite direction. The following code snippet shows how to create a quarter of a donut in the clockwise direction. The same profile circle is used, but the angle of revolution is set to -90 degrees.

[7]:
```
# Initialize the donut sketch design
design = m.create_design("quarter-donut-negative")

# Revolve the profile around the Z axis and center in the absolute origin
# for an angle of -90 degrees (clockwise)
design.revolve_sketch(
    "quarter-donut-body-negative",
    sketch=profile,
```

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```
    axis=UNITVECTOR3D_Z,
    angle=Angle(-90, unit=UNITS.degrees),
    rotation_origin=Point3D([0, 0, 0]),
)

design.plot()
```

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

Close the modeler

[8]:
```
# Close the modeler
m.close()
```

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8.3.10 Modeling: Exporting designs

After creating a design, you typically want to bring it into a CAD tool for further development. This notebook demonstrates how to export a design to the various supported CAD formats.

Create a design

The code creates a simple design for demonstration purposes. The design consists of a set of rectangular pads with a circular hole in the center.

[1]:
```
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, UNITVECTOR3D_Y, Point2D
from ansys.geometry.core.sketch import Sketch

# Instantiate the modeler
modeler = launch_modeler()

# Create a design
design = modeler.create_design("ExportDesignExample")

# Create a sketch
sketch = Sketch()
```

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```
# Create a simple rectangle
sketch.box(Point2D([0, 0]), 10, 5)

# Extrude the sketch and displace the resulting body
# to make a plane of rectangles
for x_step in [-60, -45, -30, -15, 0, 15, 30, 45, 60]:
    for y_step in [-40, -30, -20, -10, 0, 10, 20, 30, 40]:
        # Extrude the sketch
        body = design.extrude_sketch(f"Body_X_{x_step}_Y_{y_step}", sketch, 5)

        # Displace the body in the x and y directions
        body.translate(UNITVECTOR3D_X, x_step)
        body.translate(UNITVECTOR3D_Y, y_step)

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

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

Export the design

You can export the design to various CAD formats. For the formats supported see the DesignFileFormat class, which is part of the the `design` module documentation.

Nonetheless, there are a set of convenience methods that you can use to export the design to the supported formats. The following code snippets demonstrate how to do it. You can decide whether to export the design to a file in a certain directory or in the current working directory.

Export to a file in the current working directory

[2]:
```
# Export the design to a file in the current working directory
file_location = design.export_to_scdocx()
print(f"Design exported to {file_location}")
```

```
Design exported to /home/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 /home/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 /home/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 /home/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 /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↪examples/03_modeling/downloads/ExportDesignExample.xmt_bin
Does the file exist? True
```

Export to STEP format

```
# 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()}")
```

Export to IGES format

```
# 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()}")
```

Export to FMD format

[7]:
```
# 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 /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↪examples/03_modeling/downloads/ExportDesignExample.fmd
Does the file exist? True
```

Export to PMDB format

```
# Export the design to a file in the requested directory
file_location = design.export_to_pmdb(download_dir)
```

Export to Discovery format

[8]:
```
# Export the design to a file in the requested directory
file_location = design.export_to_disco(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 /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↪examples/03_modeling/downloads/ExportDesignExample.dsco
Does the file exist? True
```

Close the modeler

Close the modeler after exporting the design.

[9]:
```
# Close the modeler
modeler.close()
```

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8.3.11 Modeling: Visualization of the design tree on terminal

A user can visualize its model object tree easily by using the `tree_print()` method available on the `Design` and `Component` objects. This method prints the tree structure of the model in the terminal.

Perform required imports

For the following example, you need to import these modules:

[1]:
```
from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math.constants import UNITVECTOR3D_X, UNITVECTOR3D_Y
from ansys.geometry.core.math.point import Point2D, Point3D
from ansys.geometry.core.misc.units import UNITS
from ansys.geometry.core.sketch.sketch import Sketch
```

Create a design

The following code creates a simple design for demonstration purposes. The design consists of several cylinders extruded. The interesting part is visualizing the corresponding design tree.

[2]:
```
# Create a modeler object
modeler = launch_modeler()

# Create your design on the server side
design = modeler.create_design("TreePrintComponent")

# Create a Sketch object and draw a circle (all client side)
sketch = Sketch()
sketch.circle(Point2D([-30, -30]), 10 * UNITS.m)
distance = 30 * UNITS.m

#  The following component hierarchy is made
#
#           |---> comp_1 ---|---> nested_1_comp_1 ---> nested_1_nested_1_comp_1
#           |               |
#           |               |---> nested_2_comp_1
#           |
# DESIGN ---|---> comp_2 -------> nested_1_comp_2
#           |
#           |
#           |---> comp_3
#
#
# Now, only "comp_3", "nested_2_comp_1" and "nested_1_nested_1_comp_1"
# has a body associated.
```

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```
#

# Create the components
comp_1 = design.add_component("Component_1")
comp_2 = design.add_component("Component_2")
comp_3 = design.add_component("Component_3")
nested_1_comp_1 = comp_1.add_component("Nested_1_Component_1")
nested_1_nested_1_comp_1 = nested_1_comp_1.add_component("Nested_1_Nested_1_Component_1")
nested_2_comp_1 = comp_1.add_component("Nested_2_Component_1")
nested_1_comp_2 = comp_2.add_component("Nested_1_Component_2")

# Create the bodies
b1 = comp_3.extrude_sketch(name="comp_3_circle", sketch=sketch, distance=distance)
b2 = nested_2_comp_1.extrude_sketch(
    name="nested_2_comp_1_circle", sketch=sketch, distance=distance
)
b2.translate(UNITVECTOR3D_X, 50)
b3 = nested_1_nested_1_comp_1.extrude_sketch(
    name="nested_1_nested_1_comp_1_circle", sketch=sketch, distance=distance
)
b3.translate(UNITVECTOR3D_Y, 50)
```

```
# Create beams (in design)
circle_profile_1 = design.add_beam_circular_profile(
    "CircleProfile1", Quantity(10, UNITS.mm), Point3D([0, 0, 0]), UNITVECTOR3D_X,␣
↪UNITVECTOR3D_Y
)
beam_1 = nested_1_comp_2.create_beam(
    Point3D([9, 99, 999], UNITS.mm), Point3D([8, 88, 888], UNITS.mm), circle_profile_1
)
```

[3]:
```
# Visualize the design (optional)
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.

[4]:
```
design.tree_print()
```

```
>>> Tree print view of component 'TreePrintComponent'

Location
--------
Root component (Design)

Subtree
-------
```

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```
(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
|---(comp) Component_3
    |---(body) comp_3_circle
```

Controlling the depth of the tree

The `tree_print()` method accepts an optional argument `depth` to control the depth of the tree to be printed. The default value is `None`, which means the entire tree is printed.

[5]:
```
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`.

[6]:
```
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
```

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```
:   |---(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.

[7]:
```
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.

[8]:
```
comp_4 = design.add_component("A_Component")
design.tree_print(depth=1)
```

```
>>> Tree print view of component 'TreePrintComponent'

Location
--------
Root component (Design)

Subtree
-------
(comp) TreePrintComponent
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3
|---(comp) A_Component
```

Now, let's print the tree structure with the components sorted alphabetically.

[9]:
```
design.tree_print(depth=1, sort_keys=True)
```

```
>>> Tree print view of component 'TreePrintComponent'

Location
--------
Root component (Design)
```

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```
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.

[10]:
```
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.

[11]:
```
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.

[12]:
```
modeler.close()
```

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8.3.12 Modeling: Body color assignment and usage

In PyAnsys Geometry, a *body* represents solids or surfaces organized within the `Design` assembly. As users might be already familiar with, Ansys CAD products (like SpaceClaim, Ansys Discovery and the Geometry Service), allow to assign colors to bodies. This example shows how to assign colors to a body, retrieve their value and how to use them in the client-side visualization.

Perform required imports

Perform the required imports.

[1]:
```
import ansys.geometry.core as pyansys_geometry

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D, UNITVECTOR3D_X, UNITVECTOR3D_Y
from ansys.geometry.core.sketch import Sketch
```

Create a box sketch

Create a `Sketch` instance and insert a box sketch with a width and height of 10 in the default plane.

[2]:
```
sketch = Sketch()
sketch.box(Point2D([0, 0]), 10, 10)
```

[2]:
```
<ansys.geometry.core.sketch.sketch.Sketch at 0x7f960464c1a0>
```

Initiate design on server

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

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

Extrude the box sketch to create the matrix style design

Given the initial sketch, you can extrude it to create a matrix style design. In this example, you can create a 2x3 matrix of bodies. Each body is separated by 30 units in the X direction and 30 units in the Y direction. You have a total of 6 bodies.

[4]:
```
translate = [[0, 30, 60], [0, 30, 60]]

for r_idx, row in enumerate(translate):
    comp = design.add_component(f"Component{r_idx}")

    for b_idx, dist in enumerate(row):
        body = comp.extrude_sketch(f"Component{r_idx}_Body{b_idx}", sketch, distance=10)
        body.translate(UNITVECTOR3D_Y, r_idx*30)
        body.translate(UNITVECTOR3D_X, dist)

design.plot()
```

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

Assign colors to the bodies

Given the previous design, you can assign a color to each body. You could have done this assignment while creating the bodies, but for the sake of encapsulating the color assignment logic, it is done in its own code cell.

[5]:
```
colors = [["red", "blue", "yellow"], ["orange", "green", "purple"]]

for c_idx, comp in enumerate(design.components):
    for b_idx, body in enumerate(comp.bodies):
        body.color = colors[c_idx][b_idx]
        print(f"Body {body.name} has color {body.color}")
```

```
Body Component0_Body0 has color #ff0000ff
Body Component0_Body1 has color #0000ffff
Body Component0_Body2 has color #ffff00ff
Body Component1_Body0 has color #ffa500ff
Body Component1_Body1 has color #008000ff
Body Component1_Body2 has color #800080ff
```

Plotting the design with colors

By default, the plot method does **not** use the colors assigned to the bodies. To plot the design with the assigned colors, you need to specifically request it.

Users have two options for plotting with the assigned colors:

- Pass the parameter `use_service_colors=True` to the plot method.
- Set the global parameter `USE_SERVICE_COLORS` to `True`.

It is important to note that the usage of colors when plotting might slow down the plotting process, as it requires additional information to be sent from the server to the client and processed in the client side.

If you just request the plot without setting the global parameter, the plot will be displayed without the colors, as shown below.

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

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

As stated previously, if you pass the parameter `use_service_colors=True` to the plot method, the plot is displayed with the assigned colors.

[7]:
```
design.plot(use_service_colors=True)
```

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

However, if you set the global parameter to `True`, the plot is displayed with the assigned colors without the need to pass the parameter to the plot method.

[8]:
```
import ansys.geometry.core as pyansys_geometry

pyansys_geometry.USE_SERVICE_COLORS = True

design.plot()

# Reverting the global parameter to its default value
pyansys_geometry.USE_SERVICE_COLORS = False
```

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

This last method is useful when the user wants to plot all the designs with the assigned colors without the need to pass the parameter to the plot method in every call.

Plotting specific bodies or components with colors

If the user wants to plot specific bodies with the assigned colors, the user can follow the same approach as before. The user can pass the parameter `use_service_colors=True` to the plot method or set the global parameter `USE_SERVICE_COLORS` to `True`.

In the following examples, you are shown how to do this using the `use_service_colors=True` parameter.

Let's plot the first body of the first component with the assigned colors.

[9]:
```
body = design.components[0].bodies[0]

body.plot(use_service_colors=True)
```

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

Now, let's plot the second component with the assigned colors.

[10]:
```
comp = design.components[1]

comp.plot(use_service_colors=True)
```

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

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8.3.13 Modeling: Surface bodies and trimmed surfaces

This example shows how to trim different surfaces, and how to use those surfaces to create surface bodies.

Create a surface

Create a sphere surface. This can be done without launching the modeler.

[1]:
```
from ansys.geometry.core.shapes.surfaces import Sphere
from ansys.geometry.core.math import Point3D

surface = Sphere(origin=Point3D([0, 0, 0]), radius=1)
```

Now get information on how the surface is defined and parameterized.

[2]:
```
surface.parameterization()
```

[2]:
```
(Parameterization(form=ParamForm.PERIODIC, type=ParamType.CIRCULAR,␣
↪interval=Interval(start=0, end=6.283185307179586)),
 Parameterization(form=ParamForm.CLOSED, type=ParamType.OTHER, interval=Interval(start=-
↪1.5707963267948966, end=1.5707963267948966)))
```

Trim the surface

For a sphere, its parametization is (`u: [0, 2*pi]`, `v:[-pi/2, pi/2]`), where u corresponds to longitude and v corresponds to latitude. You can **trim** a surface by providing new parameters.

[3]:
```
from ansys.geometry.core.shapes.box_uv import BoxUV
from ansys.geometry.core.shapes.parameterization import Interval
import math

trimmed_surface = surface.trim(BoxUV(range_u=Interval(0, math.pi), range_v=Interval(0,␣
↪math.pi/2)))
```

From a `TrimmedSurface`, you can always refer back to the underlying `Surface` if needed.

[4]:
```
trimmed_surface.geometry
```

[4]:
```
<ansys.geometry.core.shapes.surfaces.sphere.Sphere at 0x7efef4ccc1a0>
```

Create a surface body

Now create a surface body by launching the modeler session and providing the trimmed surface. Then plot the body to see how you created a quarter of a sphere as a surface body.

[5]:
```
from ansys.geometry.core import launch_modeler

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

```
Ansys Geometry Modeler (0x7efef4ccea50)

Ansys Geometry Modeler Client (0x7efef4b712b0)
  Target:     localhost:700
  Connection: Healthy
```

[6]:
```
design = modeler.create_design("SurfaceBodyExample")
body = design.create_body_from_surface("trimmed_sphere", trimmed_surface)
design.plot()
```

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

If the sphere was left untrimmed, it would create a solid body since the surface is fully closed. In this case, since the surface was open, it created a surface body.

This same process can be used with other surfaces including:

- `Cone`
- `Cylinder`
- `Plane`
- `Torus`

Each surface has its own unique parameterization, which must be understood before trying to trim it.

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

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

Download this example

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

Download this example

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8.3.14 Modeling: Using design parameters

You can read and update parameters that are part of the design. The simple design in this example has two associated parameters.

Perform required imports

[1]:
```
from pathlib import Path
import requests

from ansys.geometry.core import launch_modeler
```

The file for this example is in the integration tests folder and can be downloaded.

Download the example file

Download the file for this example from the integration tests folder in the PyAnsys Geometry repository.

[2]:
```
def download_file(url, filename):
    """Download a file from a URL and save it to a local file."""
    response = requests.get(url)
    response.raise_for_status()  # Check if the request was successful
    with open(filename, 'wb') as file:
        file.write(response.content)

# URL of the file to download
url = "https://raw.githubusercontent.com/ansys/pyansys-geometry/main/tests/integration/
↪files/blockswithparameters.dsco"

# Local path to save the file to
file_path = Path.cwd() / "blockswithparameters.dsco"

# Download the file
download_file(url, file_path)
print(f"File is downloaded to {file_path}")
```

```
File is downloaded to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↪examples/03_modeling/blockswithparameters.dsco
```

Import a design with parameters

Import the model using the `open_file()` method of the modeler.

[3]:
```
# Create a modeler object
modeler = launch_modeler()
design = modeler.open_file(file_path)
design.plot()
```

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

Read existing parameters of the design

You can get all the parameters of the design as a list of parameters. Because this example has two parameters, you see two items in the list.

[4]:
```
my_parameters = design.parameters
print(len(my_parameters))
```

```
2
```

A parameter object has a name, value, and unit.

[5]:
```
print(my_parameters[0].name)
print(my_parameters[0].dimension_value)
print(my_parameters[0].dimension_type)

print(my_parameters[1].name)
print(my_parameters[1].dimension_value)
print(my_parameters[1].dimension_type)
```

```
p1
0.00010872999999999982
ParameterType.DIMENSIONTYPE_AREA
p2
0.0002552758322160814
ParameterType.DIMENSIONTYPE_AREA
```

Parameter values are returned in the default unit for each dimension type. Since default length unit is meter and default area unit is meter square, the value is returned in square meters.

Edit a parameter value

You can edit the parameter's name or value by simply setting these fields. Set the second parameter (p2 value to 350 mm).

[6]:
```
parameter1 = my_parameters[1]
parameter1.dimension_value = 0.000440
response = design.set_parameter(parameter1)
print(response)
print(my_parameters[0].dimension_value)
print(my_parameters[1].dimension_value)
```

```
ParameterUpdateStatus.SUCCESS
0.00010872999999999982
0.00044
```

After a successful parameter update, the design is updated. If we request the design plot again, we see the updated design.

[7]:
```
design.plot()
```

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

The `set_parameter()` method returns a `Success` status message if the parameter is updated or a "FAILURE" status message if the update fails. If the `p2` parameter depends on the `p1` parameter, updating the `p1` parameter might also change the `p2` parameter. In such cases, the method returns `CONSTRAINED_PARAMETERS`, which indicates other parameters were also updated.

[8]:
```
parameter1 = my_parameters[0]
parameter1.dimension_value = 0.000250
response = design.set_parameter(parameter1)
print(response)
```

```
ParameterUpdateStatus.CONSTRAINED_PARAMETERS
```

To get the updated list, query the parameters once again.

[9]:
```
my_parameters = design.parameters
print(my_parameters[0].dimension_value)
print(my_parameters[1].dimension_value)
```

```
0.00024999999999997263
0.0005812699999999444
```

Close the modeler

Close the modeler to free up resources and release the connection.

[10]:
```
modeler.close()
```

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Download this example

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8.3.15 Modeling: Chamfer edges and faces

A chamfer is an angled cut on an edge. Chamfers can be created using the `Modeler.geometry_commands` module.

Create a block

Launch the modeler and create a block.

[1]:
```
from ansys.geometry.core import launch_modeler, Modeler

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

```
Ansys Geometry Modeler (0x7fcc926152b0)

Ansys Geometry Modeler Client (0x7fcc92617a10)
  Target:     localhost:700
  Connection: Healthy
```

[2]:
```
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import Point2D

design = modeler.create_design("chamfer_block")
body = design.extrude_sketch("block", Sketch().box(Point2D([0, 0]), 1, 1), 1)

body.plot()
```

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

Chamfer edges

Create a uniform chamfer on all edges of the block.

[3]:
```
modeler.geometry_commands.chamfer(body.edges, distance=0.1)

body.plot()
```

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

Chamfer faces

The chamfer of a face can also be modified. Create a chamfer on a single edge and then modify the chamfer distance value by providing the newly created face that represents the chamfer.

[4]:
```
body = design.extrude_sketch("box", Sketch().box(Point2D([0,0]), 1, 1), 1)

modeler.geometry_commands.chamfer(body.edges[0], distance=0.1)

body.plot()
```

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

[5]:
```
modeler.geometry_commands.chamfer(body.faces[-1], distance=0.3)

body.plot()
```

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

Download this example

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

8.4 Applied examples

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

Download this example

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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]:
```
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.
```

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```
    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
    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)
```

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```
        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}")

# 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 /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/examples/
↪04_applied/NACA_Airfoil_0012.fmd
```

Close the modeler

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

Example of a cambered airfoil: NACA 6412

This code creates a NACA 6412 airfoil, which is cambered. This airfoil has a maximum camber of 6% and a maximum thickness of 12%. After overriding the NACA number, the code generates the airfoil points.

[8]:
```
NACA_AIRFOIL = "6412"
```

Generate the airfoil points

As before, you generate the points that define the NACA 6412 airfoil. Create a `Sketch` object and add the points to it. Then, approximate the airfoil using straight lines.

[9]:
```
# Create a sketch
sketch = Sketch()

# Generate the points of the airfoil
points = naca_airfoil_4digits(NACA_AIRFOIL)

# Create the segments of the airfoil
for i in range(len(points) - 1):
    sketch.segment(points[i], points[i + 1])

# Close the airfoil
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 /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/examples/
↪04_applied/NACA_Airfoil_6412.scdocx
```

Close the modeler

[12]:
```
modeler.close()
```

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8.4.2 Applied: Prepare a NACA airfoil for a Fluent simulation

Once a NACA airfoil is designed, it is necessary to prepare the geometry for a CFD simulation. This notebook demonstrates how to prepare a NACA 6412 airfoil for a Fluent simulation. The starting point of this example is the previously designed NACA 6412 airfoil. The airfoil was saved in an SCDOCX file, which is now imported into the notebook. The geometry is then prepared for the simulation.

In case you want to run this notebook, make sure that you have run the previous notebook to design the NACA 6412 airfoil.

Import the NACA 6412 airfoil

The following code starts up the Geometry Service and imports the NACA 6412 airfoil. The airfoil is then displayed in the notebook.

[1]:
```
import os

from ansys.geometry.core import launch_modeler

# Launch the modeler
modeler = launch_modeler()

# Import the NACA 6412 airfoil
design = modeler.open_file(os.path.join(os.getcwd(), f"NACA_Airfoil_6412.scdocx"))

# Retrieve the airfoil body
airfoil = design.bodies[0]

# Display the airfoil
design.plot()
```

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

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```
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 0x7fc9d8150180
  Name                 : Airfoil Faces
  Id                   : 1:72
  N Bodies             : 0
  N Faces              : 399
  N Edges              : 0
  N Beams              : 0
  N Design Points      : 0
  N Components         : 0
  N Vertices           : 0
```

Display the geometry

The geometry is now ready for the simulation. The following code displays the geometry in the notebook. This example uses the `GeometryPlotter` class to display the geometry for the airfoil and fluid domain in different colors with a specified opacity level.

The airfoil is displayed in green, and the fluid domain is displayed in blue with an opacity of 0.25.

[4]:
```
from ansys.geometry.core.plotting import GeometryPlotter

plotter = GeometryPlotter()
plotter.plot(airfoil, color="green")
plotter.plot(fluid, color="blue", opacity=0.15)
plotter.show()
```

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

Export the geometry

Export the geometry into a Fluent-compatible format. The following code exports the geometry into a PMDB file, which retains the named selections.

```
# Save the design
file = design.export_to_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

[5]:
```
modeler.close()
```

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8.4.3 Applied: Develop an external aerodynamic simulation model for Fluent analysis

The Ahmed body is a simplified car model used to study airflow around vehicles. The wake (turbulent flow behind the body) depends on the slant angle:

- Less than 12 degrees: The airflow stays attached to the slant, creating low drag and a mostly two-dimensional flow.
- 12 to 30 degrees: The flow becomes three-dimensional with strong c-pillar vortices, peaking at 30 degrees. This increases drag due to low-pressure areas on the rear surfaces.
- More than 30 degrees: The flow fully separates from the slant, reducing drag and weakening the c-pillar vortices.

This example creates an Ahmed body with a slant angle of 20 degrees. It consists of these steps:

1. Launch PyAnsys Geometry and define the default units.
2. Create sketches for the Ahmed body, enclosure, and BOI (Body of Influence).
3. Generate solid bodies from the sketches.
4. Perform Boolean operations for region extraction.
5. Group faces and define a named selection.
6. Export model as a CAD file.
7. Close session.

Define function for sorting planar face pairs along any axis

This function is used to sort the planar faces along any of the coordinate axis. This is used primarily for sorting the faces to define the named selections.

[1]:
```
def face_identifier(faces, axis):
    """
    Sort a pair of planar faces based on their positions along the specified coordinate␣
↪axis.

    Args:
        faces : List[IFace, IFace]
        List of planar face pairs.

        axis : (string)
        Axis to sort the face pair on. Options are "x", "y", or "z".
```

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```
    Returns:
        IFace, IFace
        - IFace: Face with the centroid positioned behind the other face along the␣
↪specified axis.
        - IFace: Face with the centroid positioned ahead of the other face along the␣
↪specified axis.

    """
    min_face = ""
    max_face = ""
    if axis == "x":
        position = 0
    elif axis == "y":
        position = 1
    else:
        position = 2
    min_face_cor_val = faces[0].point(0.5, 0.5)[position]
    min_face = faces[0]
    max_face_cor_val = faces[0].point(0.5, 0.5)[position]
    max_face = faces[0]
    for face in faces[1:]:
        if face.point(0.5, 0.5)[position] < min_face_cor_val:
            min_face_cor_val = face.point(0.5, 0.5)[position]
            min_face = face
            continue
        elif face.point(0.5, 0.5)[position] > max_face_cor_val:
            max_face_cor_val = face.point(0.5, 0.5)[position]
            max_face = face
    return min_face, max_face
```

Define function for calculating the vertical and horizontal distances based on the slant angle

[2]:
```
def distance_calculator(hypo, slant_angle):
    """
    Calculate the horizontal and vertical distances based on the hypotenuse and slant␣
↪angle.

    Args:
        hypo : int
        Length of the hypotenuse in millimeters.

        slant_angle : int
        Slant angle in degrees.

    Returns:
        slant_x (float): Horizontal distance calculated using the sine of the slant␣
↪angle.
        slant_y (float): Vertical distance calculated using the cosine of the slant␣
↪angle.
```

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```
    """
    slant_x = hypo * math.cos(math.radians(slant_angle))
    slant_y = hypo * math.sin(math.radians(slant_angle))
    return slant_y, slant_x
```

Launch PyAnsys geometry and define the default units

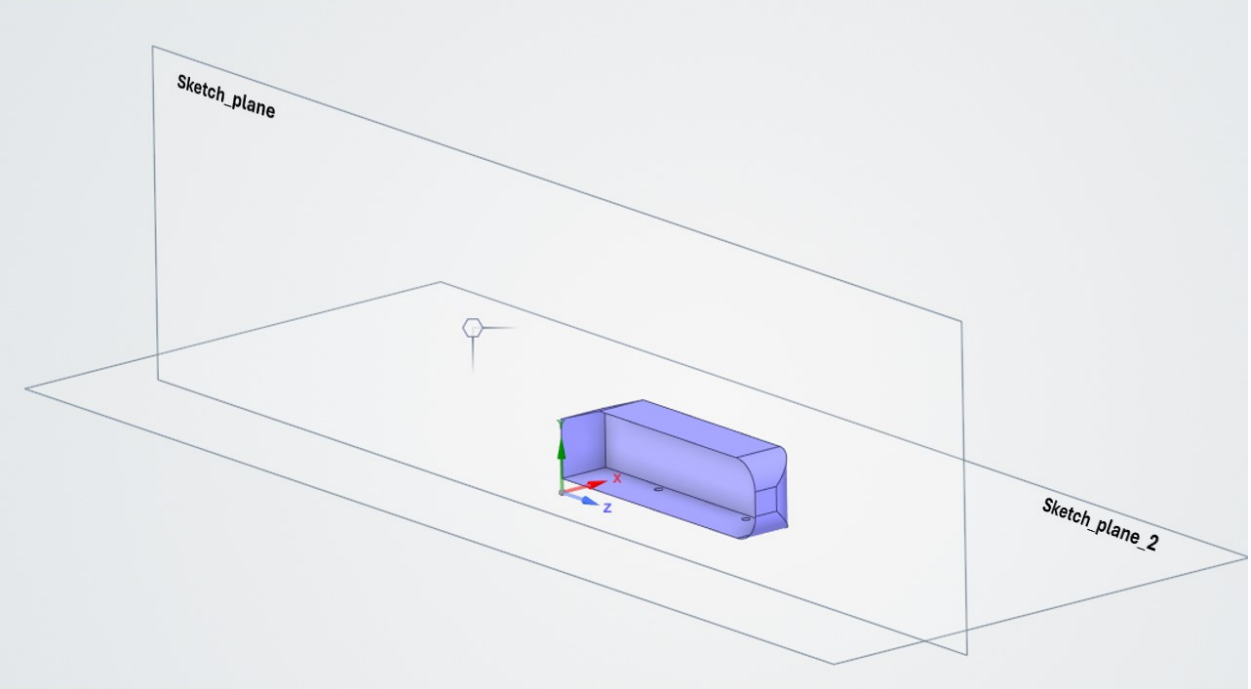
Before you start creating the Ahmed body, you must import the necessary modules to create the model using PyAnsys Geometry. It's also a good practice to define the units before initiating the development of the sketch.

[3]:
```
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import (
    Point2D,
    Plane,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.misc import UNITS, DEFAULT_UNITS

from ansys.geometry.core.plotting import GeometryPlotter
import math
import os

modeler = launch_modeler()
DEFAULT_UNITS.LENGTH = UNITS.mm
DEFAULT_UNITS.angle = UNITS.degrees
```

Create sketches for the Ahmed body, enclosure, and BOI

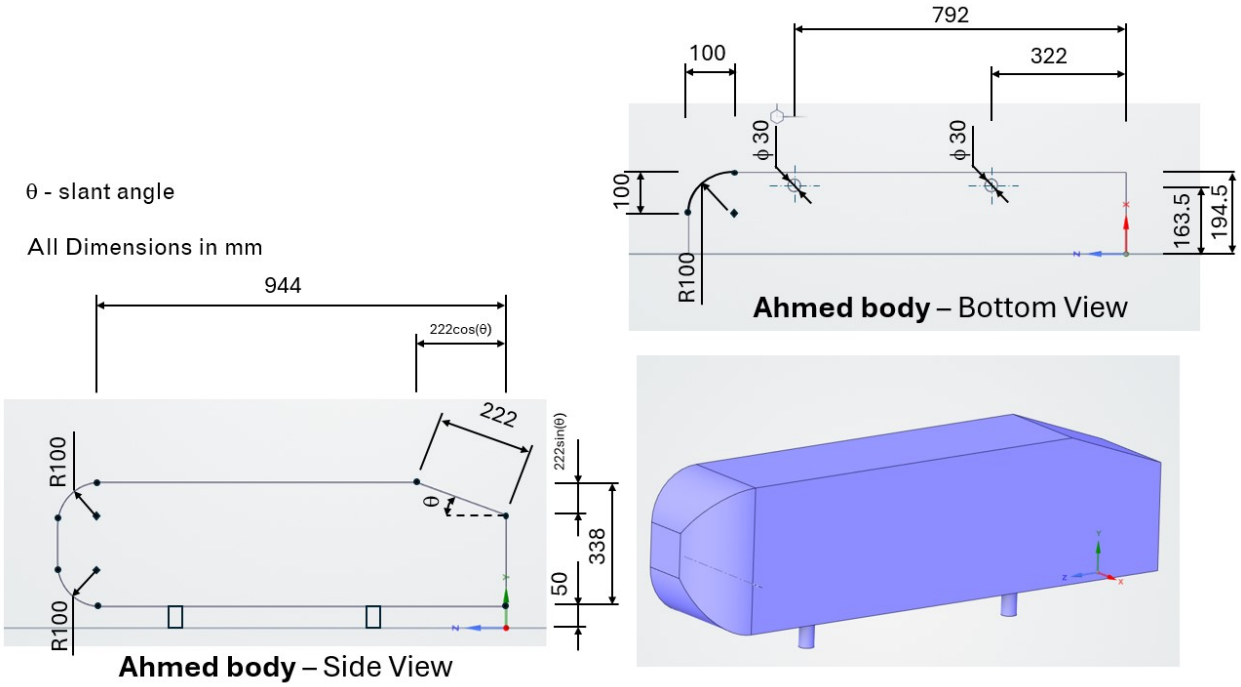
Define the appropriate sketch planes parallel to the y-z and x-z planes, passing through the origin (namely `sketch_plane` and `sketch_plane_2` respectively).

Define the sketch planes



[4]:
```
# Define sketch plane on the y-z plane passing through the origin
sketch_plane = Plane(
        origin=Point3D([0, 0, 0]),
        direction_x=UNITVECTOR3D_Y,
        direction_y=UNITVECTOR3D_Z,
    )

# Define sketch plane on the x-z plane passing through the origin
sketch_plane_2 = Plane(
    origin=Point3D([0, 0, 0]),
    direction_x=UNITVECTOR3D_X,
    direction_y=UNITVECTOR3D_Z,
)
```

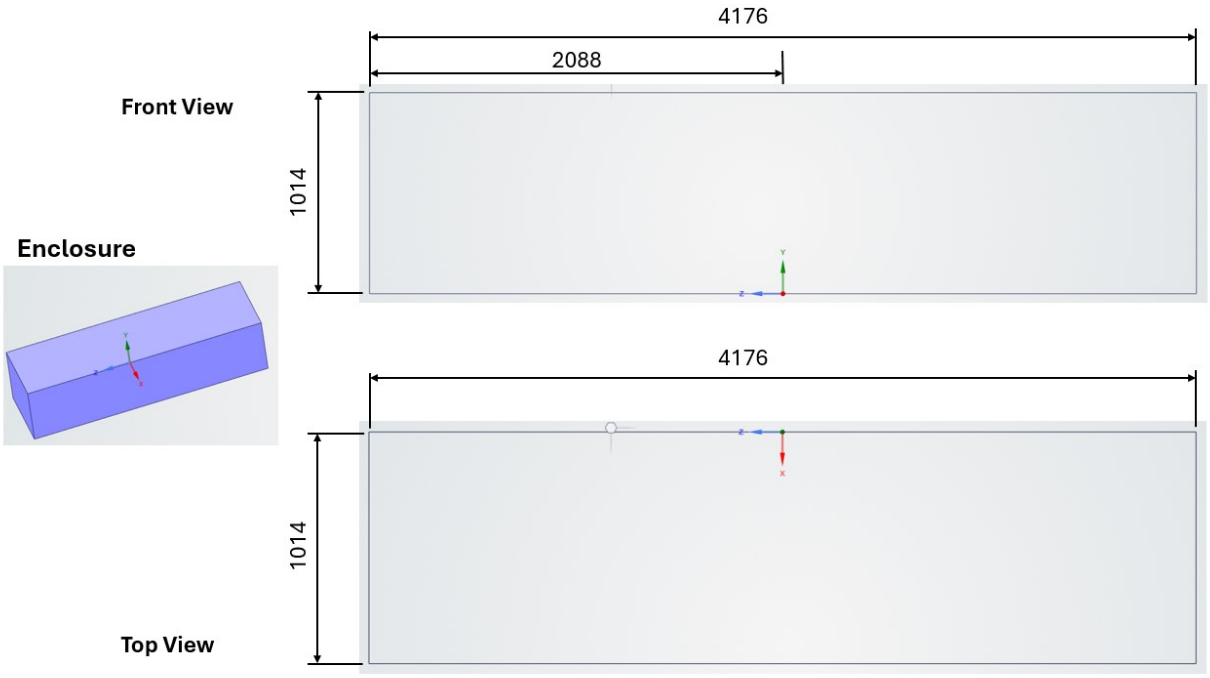
Define the Ahmed body



[5]:
```
# Calculate the horizontal and vertical distance based on slant angle of 20 degrees
slant_y, slant_x = distance_calculator(hypo=222, slant_angle=20)

# Define sketch for the Ahmed body
ahmed_body_sketch = Sketch(sketch_plane)
ahmed_body_sketch.segment(
    start=Point2D([50, 0]), end=Point2D([338 - slant_y, 0])
).segment_to_point(Point2D([338, slant_x])).segment_to_point(
    Point2D([338, 944])
).arc_to_point(
    end=Point2D([238, 1044]), center=Point2D([238, 944]), clockwise=False
).segment_to_point(
    Point2D([150, 1044])
).arc_to_point(
    end=Point2D([50, 944]), center=Point2D([150, 944]), clockwise=False
).segment_to_point(
    end=Point2D([50, 0])
)
ahmed_body_sketch.plot()
```

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

Define the enclosure

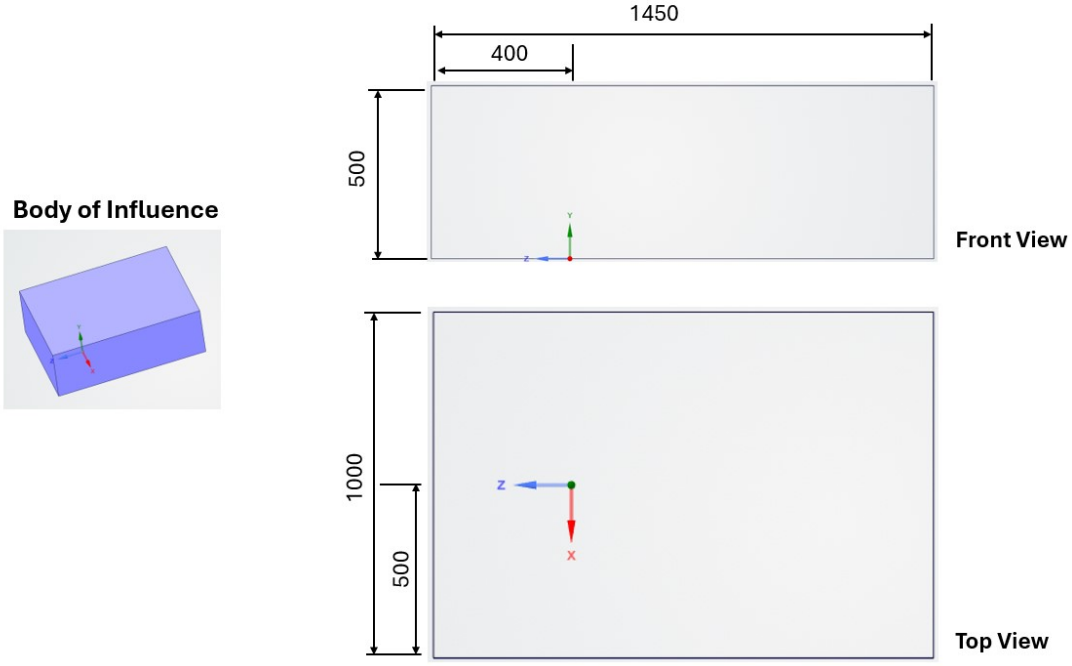


[6]:
```
# Define sketch for enclosure
enclosure_sketch = Sketch(plane=sketch_plane)
enclosure_sketch.box(center=Point2D([1014 / 2, 0]), height=4176, width=1014)
enclosure_sketch.plot()

# Define sketch for mounting 1
mount_sketch_1 = Sketch(sketch_plane_2)
mount_sketch_1.circle(center=Point2D([163.5, 792]), radius=15)

# Define sketch for mounting 2
mount_sketch_2 = Sketch(sketch_plane_2)
mount_sketch_2.circle(center=Point2D([163.5, 322]), radius=15)

# Define sketch for the fillet
ahmed_body_fillet_sketch = Sketch(sketch_plane_2)
ahmed_body_fillet_sketch.segment(
    start=Point2D([194.5, 944]), end=Point2D([194.5, 1044])
).segment_to_point(Point2D([94.5, 1044])).arc_to_point(
    Point2D([194.5, 944]), center=Point2D([94.5, 944]), clockwise=True
)
```

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

[6]:
```
<ansys.geometry.core.sketch.sketch.Sketch at 0x7f1a4ceb6ea0>
```

Define the BOI



[7]:
```
# Define sketch for BOI
boi_sketch = Sketch(sketch_plane_2)
boi_sketch.box(center=Point2D([0, -325]), width=1000, height=1450)
boi_sketch.plot()
```

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

Generate solid bodies from the sketches

From the 2D sketches, generate 3D models by extruding the sketch. First create the design body (namely `ahmed_model`, which is the root part. A component named `Component1` is created under the root part. All the bodies generated as a part of sketch extrusion would be placed within `Component1`.

[8]:
```
# Create design object
design = modeler.create_design("ahmed_model")

# Create component
component_1 = design.add_component("Component1")

# Create body `ahmed_body` by extrusion
ahmed_body = component_1.extrude_sketch(
    "ahmed_body", sketch=ahmed_body_sketch, distance=194.5
)

# Create body `ahmed_body_fillet` by cut extrusion
ahmed_body_fillet = component_1.extrude_sketch(
    "ahmed_body_fillet", sketch=ahmed_body_fillet_sketch, distance=-500, cut=True
)
```

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```
# Create body `enclosure` by extrusion
enclosure = component_1.extrude_sketch(
    "Solid1", sketch=enclosure_sketch, distance=1167
)

# Create body `mounting_1` by extrusion
mount_1 = component_1.extrude_sketch(
    "mount_1", sketch=mount_sketch_1, distance=-100
)

# Create body `mounting_2` by extrusion
mount_2 = component_1.extrude_sketch(
    "mount_2", sketch=mount_sketch_2, distance=-100
)

# Create body `boi` by extrusion
# The direction is negative since the sketch is created in X-Z plane, resulting in the␣
↪direction of normal to be parallel to the -Y axis.
boi_body = design.extrude_sketch(
    "boi", sketch=boi_sketch, distance=500, direction="-"
)
```

Perform Boolean operations for region extraction

[9]:
```
enclosure.subtract([ahmed_body, mount_1, mount_2])
enclosure.plot()
```

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

Group faces and define named selection

[10]:
```
plane_surface = []
cylindrical_surface = []

# Group faces of enclosure based on topology
for face in enclosure.faces:
    if face.surface_type.name == "SURFACETYPE_PLANE":
        plane_surface.append(face)
    elif face.surface_type.name == "SURFACETYPE_CYLINDER":
        cylindrical_surface.append(face)

wall_mount = []

# Identify faces associated with mounting
for cyl_face in cylindrical_surface:
    if cyl_face.point(0, 0.5)[1] < 0.050:
        wall_mount.append(cyl_face)

# Identify faces associated with enclosure extremes
outlet_face, inlet_face = face_identifier(faces=plane_surface, axis="z")
```

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```
symmetry_face, symmetry_x_pos = face_identifier(faces=plane_surface, axis="x")
ground, top = face_identifier(faces=plane_surface, axis="y")


# Create named selection
design.create_named_selection("wall_mount", faces=wall_mount)
design.create_named_selection("inlet", faces=[inlet_face])
design.create_named_selection("outlet", faces=[outlet_face])
design.create_named_selection("wall_ground", faces=[ground])
design.create_named_selection("symmetry_top", faces=[top])
design.create_named_selection("symmetry_center_plane", faces=[symmetry_face])
design.create_named_selection("symmetry_x_pos", faces=[symmetry_x_pos])
design.create_named_selection(
    "ahmed_body_20_0degree_boi_half-boi", faces=boi_body.faces
)

body_planar_surface = list(
    set(plane_surface)
    - set([outlet_face, inlet_face, symmetry_face, symmetry_x_pos, ground, top])
)
body_circular_surface = list(set(cylindrical_surface) - set(wall_mount))

rear_surface, front_surface = face_identifier(faces=body_planar_surface, axis="z")
bottom_surface, top_surface = face_identifier(faces=body_planar_surface, axis="y")
side_suface, symmetry_surface = face_identifier(faces=body_planar_surface, axis="x")

body_circular_surface.append(front_surface)

design.create_named_selection("wall_ahmed_body_front", faces=body_circular_surface)
design.create_named_selection(
    "wall_ahmed_body_main",
    faces=[symmetry_surface, bottom_surface, top_surface],
)

# Identify the face that forms a 20-degree angle with the y-axis.
for face in body_planar_surface:
    if round(math.degrees(math.acos(abs(face.normal().y)))) == 20:
        hypo_face = face
design.create_named_selection(
    "wall_ahmed_body_rear", faces=[hypo_face, rear_surface]
)
```

[10]:
```
ansys.geometry.core.designer.selection.NamedSelection 0x7f1a4cdf96d0
  Name                 : wall_ahmed_body_rear
  Id                   : 0:1201
  N Bodies             : 0
  N Faces              : 2
  N Edges              : 0
  N Beams              : 0
  N Design Points      : 0
  N Components         : 0
  N Vertices           : 0
```

Export model as a PMDB file

Export the geometry into a Fluent-compatible format. The following code exports the geometry into a PMDB file, which retains the named selections.

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

```
Design saved to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/examples/
↪04_applied/ahmed_model.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 Ahmed Body External Aerodynamics Simulation example in the Fluent documentation.

Close session

[12]:
```
modeler.close()
```

References

[1] S.R. Ahmed, G. Ramm, Some Salient Features of the Time-Averaged Ground Vehicle Wake,SAE-Paper 840300,1984

Download this example

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

8.5 Miscellaneous examples

Download this example

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

8.5.1 Miscellaneous: Example template

This example serves as a template for creating new examples in the documentation. It shows developers how to structure their code and comments for clarity and consistency. It also provides a basic outline for importing necessary modules, initializing the modeler, performing operations, and closing the modeler.

It is important to follow the conventions and formatting used in this example to ensure that the documentation is easy to read and understandable.

Example imports

Perform the required imports for this example. This section should include all necessary imports for the example to run correctly.

[1]:
```
# Imports
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D
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 (0x7f0d1e2012b0)

Ansys Geometry Modeler Client (0x7f0d1e203a10)
  Target:     localhost:700
  Connection: Healthy
```

Body of your example

Developers can add their code here to perform the desired operations. This section should include comments and explanations to explain what the code is doing.

Example section: Initialize a design

Create a design named `example-design`.

[3]:
```
# Initialize the example design
design = m.create_design("example-design")
```

Example section: Include images

This section demonstrates how to include static images in the documentation. You should place these images in the `doc/source/_static/` directory.

You can then reference images in the documentation using the following syntax:

Example section: Create a sketch and plot it

This section demonstrates how to create a sketch and plot it.

[4]:
```
sketch = Sketch()
sketch.box(Point2D([0, 0]), 10, 10)
sketch.plot()
```

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

Example section: Extrude the sketch and create a body

This section demonstrates how to extrude the sketch and create a body.

[5]:
```
design.extrude_sketch(f"BoxBody", sketch, distance=10)
design.plot()
```

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

Close the modeler

[6]:
```
# Close the modeler
m.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 Adding examples

Users can collaborate with examples to this documentation by adding new examples. A reference commit of the changes that adding an example requires is shown here:

https://github.com/ansys/pyansys-geometry/pull/1454/commits/7fcf02f86f05e0e5ce1c1071c3c5fcd274ec481c

To add a new example, follow these steps:

1. Create a new notebook in the `doc/source/examples` directory, under the appropriate folder for your example.
2. Use the `doc\source\examples\99_misc\template.mystnb` file as a reference for creating your example notebook. It contains the necessary metadata and structure for a PyAnsys Geometry example.
3. Add the new notebook to the `doc/source/examples.rst` file.
4. Store a thumbnail image of the example in the `doc/source/_static/thumbnails` directory.
5. Link the thumbnail image to your example file in the `doc/source/conf.py` file as shown in the reference commit.

You can also test the correct build process of a new example by performing the following steps:

1. Run the following command to install the documentation dependencies:

   ```
   pip install -e .[doc]
   ```

2. Navigate to the `doc` directory and run the following command:

   ```
   # On Linux or macOS
   make single-example examples/01_getting_started/01_math.mystnb

   # On Windows
   ./make.bat single-example examples/01_getting_started/01_math.mystnb
   ```

Note

The example name must be the same as the notebook name, with its path starting at the `examples` directory.

3. Check the `doc/source/_build/html` directory for the generated documentation and open the `index.html` file in your browser.

9.7 Run tests

PyAnsys Geometry uses pytest for testing.

9.7.1 Prerequisites

Prior to running the tests, you must run this command to install the test dependencies:

```
pip install -e .[tests]
```

Make sure to define the port and host of the service using the following environment variables:

```
# On Linux or macOS
export ANSRV_GEO_PORT=5000
export ANSRV_GEO_HOST=localhost

# On Windows CMD
set ANSRV_GEO_PORT=5000
set ANSRV_GEO_HOST=localhost

# On Windows PowerShell
$env:ANSRV_GEO_PORT=5000
$env:ANSRV_GEO_HOST="localhost"
```

9.7.2 Running the tests

To run the tests, navigate to the root directory of the repository and run this command:

```
pytest
```

Note

The tests require the Geometry Service to be installed and running on your machine. The tests fail if the service is not running. It is expected for the Geometry Service to be running as a Docker container.

If you do not have the Geometry Service running as a Docker container, but you have it running on your machine, you can still run the tests with the following argument:

```
pytest --use-existing-service=yes
```

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 3.13. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

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

```
unzip ansys-geometry-core-v0.11.0-all-wheelhouse-ubuntu-latest-3.10.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

If you are on Windows with Python 3.10, unzip to a wheelhouse directory by running `-d wheelhouse` (this is required for unzipping to a directory on Windows) and install using the preceding command.

Consider installing using a virtual environment.

The following wheelhouse files are available for download:

10.2.1 Linux

- Linux wheelhouse for Python 3.10
- Linux wheelhouse for Python 3.11
- Linux wheelhouse for Python 3.12
- Linux wheelhouse for Python 3.13

10.2.2 Windows

- Windows wheelhouse for Python 3.10
- Windows wheelhouse for Python 3.11
- Windows wheelhouse for Python 3.12
- Windows wheelhouse for Python 3.13

10.2.3 MacOS

- MacOS wheelhouse for Python 3.10
- MacOS wheelhouse for Python 3.11
- MacOS wheelhouse for Python 3.12
- MacOS wheelhouse for Python 3.13

10.3 Geometry service Docker container assets

Build the latest Geometry service Docker container using the following assets. For information on how to build the container, see *Docker containers*.

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

10.3.1 Windows container

Note

Only Ansys employees 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 0.11.0 - July 15, 2025

Added

Update with delta	#1922
Find and fix stitch/missing/small faces enhancements	#1953
Nurbs and trimmedcurve enhancements	#1994
Allow logos linux 26 1	#2048
Nurbscurve conversions	#2053
Add components to ns	#2068
Add mating conditions (align, tangent, orient)	#2069
Vertex references	#2083

Dependencies

Bump jupytext from 1.17.1 to 1.17.2 in the docs-deps group	#2024
Bump ansys-tools-visualization-interface from 0.9.1 to 0.9.2	#2027
Bump trame-vtk from 2.8.15 to 2.8.17	#2028
Bump pytest from 8.3.5 to 8.4.0	#2034
Bump requests from 2.32.3 to 2.32.4	#2040
Bump beartype from 0.20.2 to 0.21.0	#2042
Bump panel from 1.6.1 to 1.7.1	#2043
Bump ansys-tools-path from 0.7.1 to 0.7.2	#2044
Bump quarto-cli from 1.7.31 to 1.7.32	#2050
Bump ansys-tools-path from 0.7.2 to 0.7.3	#2051
Bump pytest-cov from 6.1.1 to 6.2.1	#2052
Limiting ansys-tools-visualization-interface	#2054
Bump pytest from 8.4.0 to 8.4.1	#2065
Bump panel from 1.7.1 to 1.7.2	#2077
Bump trame-vtk from 2.8.17 to 2.9.0	#2078
Bump numpydoc from 1.8.0 to 1.9.0 in the docs-deps group	#2080
Bump ansys-api-geometry from 0.4.62 to 0.4.64	#2081
Bump ansys-api-geometry from 0.4.64 to 0.4.65	#2085
Bump notebook from 7.4.3 to 7.4.4 in the docs-deps group	#2086
Bump ansys-sphinx-theme[autoapi] from 1.5.2 to 1.5.3 in the docs-deps group	#2089
Bump panel from 1.7.2 to 1.7.4	#2112
Bump pytest-pyvista from 0.1.9 to 0.2.0	#2113

Documentation

Adding extra line	#2026
Add proper disclaimer to binaries repository	#2060
Add warning section for minimum version on methods	#2062
Add deepwiki link	#2073

Fixed

Make sure export_glb is handling a single polydata object	#2032
Prevent the creation of empty named selections	#2063
Revert visualization changes	#2084
Internalize document after insert: update test	#2092

Maintenance

Update changelog for v0.10.9	#2023
Bump ansys/actions from 10.0.4 to 10.0.6 in the actions group	#2025
Bump ansys/actions from 10.0.6 to 10.0.8 in the actions group	#2029
Pre-commit automatic update	#2033, #2066, #2076, #2090, #2111
Bump ansys/actions from 10.0.8 to 10.0.9 in the actions group	#2035
Bump ansys/actions from 10.0.9 to 10.0.10 in the actions group	#2038
Bump the actions group with 2 updates	#2041
Upload code coverage on linux	#2049
Bump ansys/actions from 10.0.11 to 10.0.12 in the actions group	#2071
Bump github/codeql-action from 3.29.0 to 3.29.1 in the actions group	#2075
Bump github/codeql-action from 3.29.1 to 3.29.2 in the actions group	#2079
Using proper artifactory location	#2114

Test

Expand code coverage and fix a few things	#2039
Add more tests and update some tests	#2046
Expanding test coverage for sketch	#2047
Expanding test coverage for designer and math	#2061
Adding test coverage for designer, sketch, misc	#2070
Add more tests to expand coverage	#2087
Add stride named selection import test	#2088
Add more code coverage	#2096
Logo removal should work on linux now	#2098
Expand coverage and add bug fix test	#2103
Bug fix test and round trip open file tests	#2107

11.2 0.10.9 - June 05, 2025

Dependencies

bump ansys-sphinx-theme[autoapi] from 1.4.4 to 1.4.5 in the docs-deps group	#2008
bump ansys-sphinx-theme[autoapi] from 1.4.5 to 1.5.0 in the docs-deps group	#2009
bump notebook from 7.4.2 to 7.4.3 in the docs-deps group	#2010
bump geomdl from 5.3.1 to 5.4.0	#2012
bump ansys-sphinx-theme[autoapi] from 1.5.0 to 1.5.2 in the docs-deps group	#2014

Fixed

Typo in the open request construction	#2022

Maintenance

update CHANGELOG for v0.10.8	#2006
bump ansys/actions from 9.0.12 to 9.0.13 in the actions group	#2011
pre-commit automatic update	#2015
improving CodeQL	#2016
fix labeler permissions	#2017
Bump ansys/actions from 9.0.13 to 10.0.1 in the actions group	#2018
Bump the actions group with 2 updates	#2019, #2021

Test

Update Reader support info and add more import tests	#2013

11.3 0.10.8 - May 27, 2025

Added

repair tools refactoring	#1912
use license metadata properly	#1961
add 261 version api versions list	#1980
grpc reachitecture - several modules	#1988
deprecating `product_version` in favor of `version`	#1998

Dependencies

bump quarto-cli from 1.6.42 to 1.7.29	#1962
bump jupytext from 1.16.7 to 1.17.1 in the docs-deps group	#1963
bump ansys-sphinx-theme[autoapi] from 1.4.2 to 1.4.3 in the docs-deps group	#1967
bump ansys-api-geometry from 0.4.58 to 0.4.59	#1968
bump ansys-sphinx-theme[autoapi] from 1.4.3 to 1.4.4 in the docs-deps group	#1969
bump notebook from 7.4.1 to 7.4.2 in the docs-deps group	#1971
bump quarto-cli from 1.7.29 to 1.7.30	#1972
bump matplotlib from 3.10.1 to 3.10.3	#1974
bump pyvista[jupyter] from 0.45.0 to 0.45.1	#1975
bump scipy from 1.15.2 to 1.15.3	#1976
bump pyvista[jupyter] from 0.45.1 to 0.45.2	#1981
bump ansys-api-geometry from 0.4.59 to 0.4.60	#1987
bump quarto-cli from 1.7.30 to 1.7.31	#1991
bump ansys-api-geometry from 0.4.60 to 0.4.61	#1992
bump numpy from 2.2.5 to 2.2.6	#1995
bump ansys-api-geometry from 0.4.61 to 0.4.62	#2003

Documentation

change python3statement url	#1965

Fixed

myst warning – all cells must be of same type	#1970

Maintenance

update CHANGELOG for v0.10.7	#1959
pre-commit automatic update	#1964, #1978, #1993, #2004
bump ansys/actions from 9.0.7 to 9.0.8 in the actions group	#1966
bump ansys/actions from 9.0.8 to 9.0.9 in the actions group	#1973
bump ansys/actions from 9.0.9 to 9.0.19 in the actions group	#1979
bump ansys/actions from 9.0.19 to 9.0.20 in the actions group	#1982
bump ansys/actions from 9.0.20 to 9.0.21 in the actions group	#1983
bump ansys/actions from 9.0.21 to 9.0.22 in the actions group	#1984
revert ansys/actions release	#1985
bump the actions group with 2 updates	#1990
bump ansys/actions from 9.0.9 to 9.0.11 in the actions group	#1996
fix the usage of unstable container skip	#2001
bump ansys/actions from 369ef11a9888875682d1a6b0ec65f82c4d8a664d to 5dc39c7838f50142138f7ac518ff3e4dca065d97 in the actions group	#2002

11.4 0.10.7 - May 05, 2025

Added

grpc driving dimensions stub implementation	#1921
move coordinate systems stub to grpc layer	#1943

Dependencies

bump numpy from 2.2.4 to 2.2.5	#1935
bump pygltflib from 1.16.3 to 1.16.4	#1940
bump notebook from 7.3.3 to 7.4.1 in the docs-deps group	#1946
bump ansys-api-geometry from 0.4.57 to 0.4.58	#1954

Documentation

Update `CONTRIBUTORS.md` with the latest contributors	#1938
ignore stackoverflow link	#1957

Fixed

core service launcher missing CADIntegration bin folder in path	#1958

Maintenance

update CHANGELOG for v0.10.6	#1933
use v4 of pyvista/setup-headless-display-action	#1934
bump github/codeql-action from 3.28.15 to 3.28.16 in the actions group	#1936
bump the actions group with 2 updates	#1937, #1942
pre-commit automatic update	#1941
bump github/codeql-action from 3.28.16 to 3.28.17 in the actions group	#1956

11.5 0.10.6 - April 22, 2025

Added

grpc prepare tools stub implementation	#1914

Dependencies

bump PyVista and VTK versions (support Python 3.13)	#1924

Fixed

docstyle ordering	#1925
adapt Native folder path for Linux and Windows	#1932

Maintenance

update CHANGELOG for v0.10.5	#1919
bump skitionek/notify-microsoft-teams to v1.0.9 in the actions group	#1920
bump ansys/actions from 9.0.2 to 9.0.3 in the actions group	#1923
fix issues with OSMesa installation and env variables set up	#1927
bump ansys/actions from 9.0.3 to 9.0.6 in the actions group	#1928
pre-commit automatic update	#1930
fix unstable workflows for Linux (missing headless display)	#1931

11.6 0.10.5 - April 16, 2025

Added

grpc measurement tools stub implementation	#1909

Dependencies

bump ansys-api-geometry from 0.4.56 to 0.4.57	#1906
bump grpcio limits and handle erratic gRPC channel creation	#1913

Documentation

Update `CONTRIBUTORS.md` with the latest contributors	#1907

Fixed

is_suppressed is not available until 25R2	#1916

Maintenance

update CHANGELOG for v0.10.4	#1901
make doc releases dependent on GH and PyPI release	#1902
bump ansys/actions from 9.0.1 to 9.0.2 in the actions group	#1903
bump skitionek/notify-microsoft-teams from 190d4d92146df11f854709774a4dae6eaf5e2aa3 to fab6aca2609ba706ebc981d066278d47ab4af0fc in the actions group	#1910
pre-commit automatic update	#1911
bump the actions group with 2 updates	#1915
update CHANGELOG for v0.8.3	#1917
update CHANGELOG for v0.9.2	#1918

11.7 0.10.4 - April 09, 2025

Added

grpc named selection stub implementation	#1899

Dependencies

bump ansys-api-geometry from 0.4.55 to 0.4.56	#1896

Documentation

Ahmed body example for external aero simulation	#1886
adding command for single example build	#1893

Fixed

geomdl dependency in conda-forge	#1900

Maintenance

update CHANGELOG for v0.10.3	#1894
upgrading to ansys/actions v9 and securing action usage	#1895
bump the actions group with 3 updates	#1897
remove whitelisting	#1898

11.8 0.10.3 - April 08, 2025

Added

grpc common layer architecture, bodies stub and admin stub implementation	#1867
Logo detection	#1873
DbuApplication stub relocation	#1882

Dependencies

bump ansys-sphinx-theme[autoapi] from 1.3.3 to 1.4.2 in the docs-deps group	#1874
bump ansys-api-geometry from 0.4.50 to 0.4.54	#1875
bump pytest-cov from 6.0.0 to 6.1.0	#1880
bump pytest-cov from 6.1.0 to 6.1.1	#1888
bump ansys-api-geometry from 0.4.54 to 0.4.55	#1889

Documentation

Update `CONTRIBUTORS.md` with the latest contributors	#1887

Fixed

Core Service install location on official build changed	#1876

Maintenance

update CHANGELOG for v0.10.2	#1870
pre-commit automatic update	#1878, #1890

Test

issue 1868 (named selection beams testing)	#1871

11.9 0.10.2 - March 26, 2025

Added

implement lazy loading of members in NamedSelection to speed up loading times when reading model	#1869

Dependencies

bump beartype from 0.19.0 to 0.20.1	#1862
bump beartype from 0.20.1 to 0.20.2	#1864

Maintenance

update CHANGELOG for v0.10.1	#1861
pre-commit automatic update	#1866

Test

issue 1801	#1865

11.10 0.10.1 - March 21, 2025

Maintenance

update CHANGELOG for v0.10.0	#1856
bump version number to 0.11.dev0	#1857
fix release notes inputs	#1858
cleanup deprecated methods	#1860

11.11 0.10.0 - March 21, 2025

Added

named selection functionality	#1768
Streaming upload support	#1779
imprint curves without a sketch	#1781
RGBA color support	#1788
enhanced 3D bounding box implementation	#1805
matrix helper methods	#1806
component name setting	#1820
enable runscript for CoreService	#1821
enhanced beam implementation	#1828
update api geometry dependency	#1834
revolve faces and revolve faces by helix options	#1842
Remove rounds	#1851
blitz (2nd round)	#1853

Dependencies

bump matplotlib from 3.10.0 to 3.10.1	#1789
bump pytest from 8.3.4 to 8.3.5	#1791
bump ansys-api-geometry from 0.4.42 to 0.4.43	#1799
bump ansys-api-geometry from 0.4.43 to 0.4.44	#1803
bump ansys-api-geometry from 0.4.44 to 0.4.45	#1809
bump ansys-api-geometry from 0.4.45 to 0.4.46	#1814
bump pytest-xvfb from 3.0.0 to 3.1.1	#1822
bump ansys-api-geometry from 0.4.46 to 0.4.47	#1827
bump notebook from 7.3.2 to 7.3.3 in the docs-deps group	#1836
bump ansys-api-geometry from 0.4.47 to 0.4.48	#1837
ansys api geometry 0.4.49	#1840
bump numpy from 2.2.3 to 2.2.4	#1844
bump ansys-api-geometry from 0.4.48 to 0.4.49	#1845
bump ansys-api-geometry from 0.4.49 to 0.4.50	#1849

Fixed

flaky color test due to random face assignment	#1794
fix parasolid export tests with more precise backend descriptor	#1802
translating sketch issues when using a custom default unit	#1808
edge start and end were not being mapped correctly	#1816
change Core Service path to executable/DLL after renaming	#1841
tessellation options were not extended to component/face methods	#1850
named selection import test	#1854

Maintenance

update CHANGELOG for v0.9.1	#1787
pre-commit automatic update	#1792, #1810, #1846
remove DMS from pipelines and use core service images only	#1812
use ansys/action/hk-automerge-prs	#1824
upgrading to new features in ansys/actions v8.2	#1852
cleanup blitz PR	#1855

Test

Skip test due to SC bug	#1798
improve share topology test	#1804
Fix slow tests	#1832
adding inward shell	#1833

11.12 0.9.2 - April 16, 2025

11.12.1 Fixed

- is_suppressed is not available until 25R2 #1916

11.13 0.9.1 - 2025-02-28

11.13.1 Added

- offset faces set radius implementation + testing #1769
- separate graphics target #1782

11.13.2 Dependencies

- bump the docs-deps group with 2 updates #1762
- bump ansys-api-geometry from 0.4.38 to 0.4.40 #1763
- bump ansys-sphinx-theme[autoapi] from 1.3.1 to 1.3.2 in the docs-deps group #1766
- bump ansys-tools-visualization-interface from 0.8.1 to 0.8.3 #1767
- bump sphinx from 8.2.0 to 8.2.1 in the docs-deps group #1772
- bump ansys-api-geometry from 0.4.40 to 0.4.42 #1773
- temporary workaround for using trusted publisher approach #1783

11.13.3 Fixed

- allow setting max message length for send operations #1775
- typo in labeler.yml file #1776
- docker build process failing on helper script #1785

11.13.4 Maintenance

- bump dev version to 0.10.dev0 #1752
- update CHANGELOG for v0.9.0 #1760
- pre-commit automatic update #1770

11.14 0.9.0 - 2025-02-18

11.14.1 Added

- design activation changes #1707
- add contributors #1708
- Implementation of inspect & repair geometry #1712
- launch core service from envar #1716
- workflow enhancements for better tool results #1723
- add face color, round info, bring measure tools to linux #1732
- conservative approach to single design per modeler #1740
- export glb #1741
- allow plotting of individual faces #1757

11.14.2 Dependencies

- bump ansys-api-geometry from 0.4.33 to 0.4.34 #1709
- bump ansys-sphinx-theme[autoapi] from 1.2.6 to 1.2.7 in the docs-deps group #1719
- bump ansys-api-geometry from 0.4.34 to 0.4.35 #1720
- bump ansys-sphinx-theme[autoapi] from 1.2.7 to 1.3.0 in the docs-deps group #1726
- bump ansys-sphinx-theme[autoapi] from 1.3.0 to 1.3.1 in the docs-deps group #1728
- bump ansys-api-geometry from 0.4.35 to 0.4.36 #1729
- bump trame-vtk from 2.8.14 to 2.8.15 #1736
- bump jupytext from 1.16.6 to 1.16.7 in the docs-deps group #1742
- bump ansys-api-geometry from 0.4.36 to 0.4.37 #1743
- bump myst-parser from 4.0.0 to 4.0.1 in the docs-deps group #1744
- bump ansys-api-geometry from 0.4.37 to 0.4.38 #1746
- bump numpy from 2.2.2 to 2.2.3 #1747
- bump panel from 1.6.0 to 1.6.1 #1749
- bump scipy from 1.15.1 to 1.15.2 #1756

11.14.3 Documentation

- update CONTRIBUTING.md #1730

11.14.4 Fixed

- re enable fmd tests #1711
- support body mirror on linux #1714
- use sketch plane for imprint/project curves #1715
- revert boolean ops logic and hold-off on commands-based implementation (temporarily) #1725
- Linux Core Service docker file was missing a dependency #1758

11.14.5 Maintenance

- update CHANGELOG for v0.8.2 #1706
- pre-commit automatic update #1717, #1737
- update SECURITY.md versions supported #1722
- keep simba-plugin-geometry tag #1739
- enhancements to GLB export and object `plot()` methods #1750
- clean up deprecation warning for trapezoid class and add more info on deprecation #1754

11.14.6 Test

- verifying issue with empty intersect and temporal body creation #1258
- Expand pattern tests #1713
- set body name #1727

- activate 8 linux tests #1745

11.15 0.8.3 - April 16, 2025

11.15.1 Fixed

- is_suppressed is not available until 25R2 #1916

11.16 0.8.2 - 2025-01-29

11.16.1 Added

- create a fillet on an edge/face #1621
- create a full fillet between multiple faces #1623
- extrude existing faces, setup face offset relationships #1628
- interference repair tool #1633
- extrude existing edges to create surface bodies #1638
- create and modify linear patterns #1641
- body suppression state #1643
- parameters refurbished #1647
- rename object #1648
- surface body from trimmed curves #1650
- create circular and fill patterns #1653
- find fix simplify #1661
- replace face #1664
- commands for merge and intersect #1665
- revolve faces a set distance, up to another object, or by a helix #1666
- add split body and tests #1669
- enable get/set persistent ids for stride import/export #1671
- find and fix edge methods #1672
- shell methods #1673
- implementation of NURBS curves #1675
- get assigned material #1684
- matrix rotation and translation #1689
- is_core_service BackendType static method #1692
- export and download stride format #1698
- blitz development #1701

11.16.2 Dependencies

- bump ansys-tools-visualization-interface from 0.7.0 to 0.8.1 #1640
- bump ansys-api-geometry from 0.4.27 to 0.4.28 #1644
- bump sphinx-autodoc-typehints from 3.0.0 to 3.0.1 in the docs-deps group #1651
- bump ansys-api-geometry from 0.4.28 to 0.4.30 #1652
- bump protobuf from 5.28.3 to 5.29.3 in the grpc-deps group across 1 directory #1656
- bump numpy from 2.2.1 to 2.2.2 #1662
- bump ansys-api-geometry from 0.4.30 to 0.4.31 #1663
- bump ansys api geometry from 0.4.30 to 0.4.32 #1677
- bump ansys-api-geometry from 0.4.31 to 0.4.32 #1681
- bump panel from 1.5.5 to 1.6.0 #1682
- bump semver from 3.0.2 to 3.0.4 #1687
- bump ansys-api-geometry from 0.4.32 to 0.4.33 #1695
- bump nbconvert from 7.16.5 to 7.16.6 in the docs-deps group #1700

11.16.3 Fixed

- reactivate test on failing extra edges test #1396
- filter set export id to only CoreService based backends #1685
- cleanup unsupported module #1690
- disable unimplemented tests #1691
- tech review fixes for blitz branch #1703

11.16.4 Maintenance

- update CHANGELOG for v0.8.1 #1639
- whitelist semver package temporarily #1657
- reverting semver package whitelist since problematic version is yanked #1659
- pre-commit automatic update #1667, #1696
- ensure design is closed on test exit #1680
- use dedicate pygeometry-ci-2 runner #1693
- remove towncrier info duplicates #1702

11.16.5 Test

- add more find and fix tests for repair tools #1645
- Add some new tests #1670
- add unit tests for 3 repair tools #1683

11.17 0.8.1 - 2025-01-15

11.17.1 Dependencies

- bump ansys-api-geometry from 0.4.26 to 0.4.27 #1634

11.17.2 Fixed

- release issues encountered #1637

11.17.3 Maintenance

- update CHANGELOG for v0.8.0 #1636

11.18 0.8.0 - 2025-01-15

11.18.1 Added

- active support for Python 3.13 #1481
- add chamfer tool #1495
- allow version input to automatically consider the nuances for the Ansys Student version #1548
- adapt health check timeout algorithm #1559
- add core service support #1571
- enable (partially) prepare and repair tools in Core Service #1580
- create launcher for core services #1587

11.18.2 Dependencies

- bump ansys-api-geometry from 0.4.16 to 0.4.17 #1547
- bump ansys-sphinx-theme[autoapi] from 1.2.1 to 1.2.2 in the docs-deps group #1549
- bump ansys-api-geometry from 0.4.17 to 0.4.18 #1550
- bump ansys-tools-visualization-interface from 0.5.0 to 0.6.0 #1554
- bump pytest from 8.3.3 to 8.3.4 #1562
- bump six from 1.16.0 to 1.17.0 #1568
- bump the docs-deps group across 1 directory with 2 updates #1570
- bump ansys-api-geometry from 0.4.18 to 0.4.20 #1574
- bump numpy from 2.1.3 to 2.2.0 #1575
- bump ansys-api-geometry from 0.4.20 to 0.4.23 #1581
- bump ansys-api-geometry from 0.4.23 to 0.4.24 #1582
- bump ansys-tools-visualization-interface from 0.6.0 to 0.6.1 #1583
- bump ansys-tools-visualization-interface from 0.6.1 to 0.6.2 #1586
- avoid the usage of attrs 24.3.0 (temporary) #1589
- bump jupytext from 1.16.4 to 1.16.5 in the docs-deps group #1590

- bump jupytext from 1.16.5 to 1.16.6 in the docs-deps group #1593
- bump panel from 1.5.4 to 1.5.5 #1595
- bump ansys-sphinx-theme[autoapi] from 1.2.3 to 1.2.4 in the docs-deps group #1597
- bump notebook from 7.3.1 to 7.3.2 in the docs-deps group #1598
- bump numpy from 2.2.0 to 2.2.1 #1599
- bump ansys-tools-path from 0.7.0 to 0.7.1 #1600
- bump nbsphinx from 0.9.5 to 0.9.6 in the docs-deps group #1602
- bump nbconvert from 7.16.4 to 7.16.5 in the docs-deps group #1609
- bump ansys-api-geometry from 0.4.24 to 0.4.25 #1610
- bump sphinx-autodoc-typehints from 2.5.0 to 3.0.0 in the docs-deps group #1611
- bump scipy from 1.14.1 to 1.15.0 #1612
- bump trame-vtk from 2.8.12 to 2.8.13 #1616
- bump trame-vtk from 2.8.13 to 2.8.14 #1617
- bump ansys-tools-visualization-interface from 0.6.2 to 0.7.0 #1619
- bump ansys-sphinx-theme[autoapi] from 1.2.4 to 1.2.6 in the docs-deps group #1624
- bump scipy from 1.15.0 to 1.15.1 #1625
- bump ansys-api-geometry from 0.4.25 to 0.4.26 #1626

11.18.3 Documentation

- Explain how to report a security issue. #1605

11.18.4 Fixed

- numpydoc warnings #1556
- vtk/pyvista issues #1584
- make_child_logger only takes 2 args. #1603
- FAQ on install #1631

11.18.5 Maintenance

- pre-commit automatic update #1366, #1552, #1561, #1588, #1601, #1615, #1630
- update CHANGELOG for v0.7.6 #1545
- change release artifacts self-hosted runner #1546
- automerge pre-commit.ci PRs #1553
- bump pyvista/setup-headless-display-action to v3 #1555
- decouple unstable image promotion #1591
- skip unnecessary stages when containers are the same #1592
- Numpy is already imported at the top of the module. #1604
- update license year using pre-commit hook #1608

11.19 0.7.6 - 2024-11-19

11.19.1 Added

- allow for some additional extrusion direction names #1534

11.19.2 Dependencies

- bump ansys-sphinx-theme[autoapi] from 1.1.7 to 1.2.0 in the docs-deps group #1520
- bump ansys-tools-visualization-interface from 0.4.7 to 0.5.0 #1521
- bump numpy from 2.1.2 to 2.1.3 #1522
- bump ansys-api-geometry from 0.4.13 to 0.4.14 #1525
- bump ansys-api-geometry from 0.4.14 to 0.4.15 #1529
- bump pint from 0.24.3 to 0.24.4 #1530
- bump trame-vtk from 2.8.11 to 2.8.12 #1531
- bump ansys-sphinx-theme[autoapi] from 1.2.0 to 1.2.1 in the docs-deps group #1535
- bump panel from 1.5.3 to 1.5.4 #1536
- bump ansys-tools-path from 0.6.0 to 0.7.0 #1537
- bump ansys-api-geometry from 0.4.15 to 0.4.16 #1538
- limit upper version on grpcio & grpcio-health-checking to 1.68 #1544

11.19.3 Documentation

- typo with the docstrings #1524
- change max header links before more dropdown #1527

11.19.4 Maintenance

- update CHANGELOG for v0.7.5 #1519
- pre-commit automatic update #1523, #1532, #1543
- bump codecov/codecov-action from 4 to 5 in the actions group #1541

11.20 0.7.5 - 2024-10-31

11.20.1 Added

- create body from surface #1454
- performance enhancements to plotter #1496
- allow picking from easy access methods #1499
- implement cut operation in extrude sketch #1510
- caching bodies to avoid unnecessary object creation #1513
- enable retrieval of service logs (via API) #1515

11.20.2 Dependencies

- bump sphinx from 8.1.0 to 8.1.3 in the docs-deps group #1479
- bump ansys-sphinx-theme[autoapi] from 1.1.4 to 1.1.5 in the docs-deps group #1482
- bump the grpc-deps group across 1 directory with 3 updates #1487
- bump ansys-sphinx-theme[autoapi] from 1.1.5 to 1.1.6 in the docs-deps group #1493
- bump trame-vtk from 2.8.10 to 2.8.11 #1494
- bump ansys-api-geometry from 0.4.11 to 0.4.12 #1502
- bump protobuf from 5.28.2 to 5.28.3 in the grpc-deps group #1505
- bump ansys-sphinx-theme[autoapi] from 1.1.6 to 1.1.7 in the docs-deps group #1506
- bump ansys-tools-visualization-interface from 0.4.6 to 0.4.7 #1507
- bump panel from 1.5.2 to 1.5.3 #1508
- bump ansys-api-geometry from 0.4.12 to 0.4.13 #1512
- bump the grpc-deps group with 2 updates #1517
- bump pytest-cov from 5.0.0 to 6.0.0 #1518

11.20.3 Documentation

- avoid having a drop down in the top navigation bar #1485
- provide information on how to build a single example #1490
- add example file to download in the test #1501
- revisit examples to make sure they are properly styled #1509
- align landing page layout with UI/UX requirements #1511

11.20.4 Fixed

- static search options #1478
- respect product_version when launching geometry service #1486

11.20.5 Maintenance

- update CHANGELOG for v0.7.4 #1476
- pre-commit automatic update #1480, #1516
- avoid linkcheck on changelog (unnecessary) #1489
- update CONTRIBUTORS #1492
- allowing new tags for Windows Core Service #1497
- simplify vulnerabilities check #1504

11.21 0.7.4 - 2024-10-11

11.21.1 Dependencies

- bump sphinx from 8.0.2 to 8.1.0 in the docs-deps group #1470

- bump ansys-api-geometry from 0.4.10 to 0.4.11 #1473
- bump ansys-sphinx-theme to v1.1.3 #1475

11.21.2 Fixed

- solving intersphinx warnings on paths #1469
- `check_input_types` not working with forward refs #1471
- `share_topology` is available on 24R2 #1472

11.21.3 Maintenance

- update CHANGELOG for v0.7.3 #1466

11.22 0.7.3 - 2024-10-09

11.22.1 Added

- use service colors in plotter (upon request) #1376
- capability to close designs (also on `modeler.exit()`) #1409
- prioritize user-defined SPACECLAIM_MODE env var #1440
- verifying Linux service also accepts colors #1451

11.22.2 Dependencies

- bump protobuf from 5.28.0 to 5.28.1 in the grpc-deps group #1424
- bump the docs-deps group with 2 updates #1425, #1436
- bump ansys-tools-visualization-interface from 0.4.3 to 0.4.4 #1426
- bump pytest from 8.3.2 to 8.3.3 #1427
- bump panel from 1.4.5 to 1.5.0 #1428
- bump protobuf from 5.28.1 to 5.28.2 in the grpc-deps group #1435
- bump the grpc-deps group with 3 updates #1442
- bump beartype from 0.18.5 to 0.19.0 #1443
- bump panel from 1.5.0 to 1.5.1 #1444
- bump ansys-sphinx-theme[autoapi] from 1.1.1 to 1.1.2 in the docs-deps group #1456
- bump ansys-api-geometry from 0.4.8 to 0.4.9 #1457
- bump numpy from 2.1.1 to 2.1.2 #1458
- bump panel from 1.5.1 to 1.5.2 #1459
- bump ansys-api-geometry from 0.4.9 to 0.4.10 #1461
- bump ansys-tools-visualization-interface from 0.4.4 to 0.4.5 #1462
- update protobuf from 5.27.2 to 5.27.5 #1464
- bump sphinx-autodoc-typehints from 2.4.4 to 2.5.0 in the docs-deps group #1465

11.22.3 Documentation

- adding cheat sheet on documentation #1433
- add captions in examples toctrees #1434

11.22.4 Fixed

- ci/cd issues on documentation build #1441
- adapt tessellate tests to new core service #1449
- rename folders on Linux docker image according to new version #1450

11.22.5 Maintenance

- update CHANGELOG for v0.7.2 #1422
- checkout LFS files from previous version to ensure upload #1423
- pre-commit automatic update #1431, #1437, #1445, #1460
- update to ansys actions v8 and docs theme (static search) #1446
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- check method implemented in Ansys actions #1448
- unstable image promotion and dependabot daily updates #1463

11.23 0.7.2 - 2024-09-11

11.23.1 Added

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

11.23.2 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

11.23.3 Documentation

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

11.23.4 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

11.23.5 Maintenance

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

11.24 0.7.1 - 2024-09-06

11.24.1 Added

- get and set body color #1357
- add `modeler.exit()` method #1375
- setting instance name during component creation #1382
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11.24.2 Dependencies

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- bump scipy from 1.14.0 to 1.14.1 #1372
- bump the grpc-deps group with 3 updates #1391
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11.24.3 Documentation

- add project logo #1405

11.24.4 Fixed

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11.24.5 Maintenance

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11.25 0.7.0 - 2024-08-13

11.25.1 Added

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11.25.2 Dependencies

- build: bump the grpc-deps group with 3 updates #1342
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11.25.3 Documentation

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11.25.4 Fixed

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11.25.5 Maintenance

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11.25.6 Miscellaneous

- chore: pre-commit automatic update #1345

11.26 0.6.6 - 2024-08-01

11.26.1 Added

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11.26.3 Fixed

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11.26.4 Dependencies

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11.26.5 Miscellaneous

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

11.27 0.6.5 - 2024-07-02

11.27.1 Changed

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

11.27.2 Fixed

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

11.27.3 Dependencies

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- build: bump scipy from 1.13.1 to 1.14.0 #1284
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11.27.4 Miscellaneous

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11.28 0.6.4 - 2024-06-24

11.28.1 Added

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

11.28.2 Changed

- chore: update CHANGELOG for v0.6.3 #1273
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11.28.3 Fixed

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11.28.4 Dependencies

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11.29 0.6.3 - 2024-06-18

11.29.1 Changed

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- build: adapting to numpy 2.x #1265
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11.29.2 Fixed

- fix: unnecessary Point3D comparison #1264
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- fix: change action order #1269

11.29.3 Dependencies

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

11.29.4 Miscellaneous

- chore: pre-commit automatic update #1267

11.30 0.6.2 - 2024-06-17

11.30.1 Added

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11.30.2 Changed

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11.30.3 Fixed

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11.31 0.6.1 - 2024-06-12

11.31.1 Added

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11.31.2 Changed

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11.31.3 Fixed

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11.31.4 Dependencies

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11.31.5 Miscellaneous

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11.32 0.6.0 - 2024-06-07

11.32.1 Added

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11.32.2 Changed

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11.32.3 Fixed

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11.32.4 Dependencies

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11.32.5 Miscellaneous

- docs: update AUTHORS #1222

11.33 0.5.6 - 2024-05-23

11.33.1 Added

- feat: add new arc constructors #1208

11.33.2 Changed

- chore: update CHANGELOG for v0.5.5 #1205

11.33.3 Dependencies

- build: bump requests from 2.31.0 to 2.32.2 #1204
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11.34 0.5.5 - 2024-05-21

11.34.1 Changed

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

11.34.2 Fixed

- fix: adapting `Arc` class constructor order to (start, end, center) #1196
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11.34.3 Dependencies

- build: bump grpcio from 1.63.0 to 1.64.0 in the grpc-deps group #1198
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11.34.4 Miscellaneous

- chore: pre-commit automatic update #1202

11.35 0.5.4 - 2024-05-15

11.35.1 Added

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

11.35.2 Changed

- chore: update CHANGELOG for v0.5.3 #1177

11.35.3 Dependencies

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11.35.4 Miscellaneous

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11.36 0.5.3 - 2024-04-29

11.36.1 Fixed

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11.37 0.5.2 - 2024-04-29

11.37.1 Added

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11.37.2 Changed

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11.37.4 Miscellaneous

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11.38 0.5.1 - 2024-04-24

11.38.1 Added

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11.38.2 Changed

- ci: adapt to vale `v3` #1129
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11.38.3 Fixed

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11.38.4 Dependencies

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11.38.5 Miscellaneous

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11.39 0.5.0 - 2024-04-17

11.39.1 Added

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