



© 2024 ANSYS, Inc. or affiliated companies
Unauthorized use, distribution, or duplication prohibited.

PyAnsys Geometry



ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Apr 29, 2024

ANSYS, Inc. and
ANSYS Europe,
Ltd. are UL
registered ISO
9001:2015
companies.

CONTENTS

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


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

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

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

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

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

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

GETTING STARTED

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

1.1 Installation

You can use `pip` to install PyAnsys Geometry.

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

1.2 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.

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

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

Connect to an existing service Connect to an existing Geometry service locally or remotely.

1.3 Compatibility with Ansys releases

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

1.4 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*.

1.5 Frequently asked questions

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

1.5.1 Docker containers

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.

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.

Go to Getting started

Windows Docker container

Contents

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

- * *Environment variables*
- * *Geometry service launcher*
- *Connect to the Geometry service*

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 XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX > GH_TOKEN.txt
```

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

Powershell

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

Windows CMD

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

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

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

Build the Geometry service Windows container

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

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

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

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

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

There are two build modes:

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

Prerequisites

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

Build from available Ansys installation

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

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

```
python build_docker_windows.py
```

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

```
docker images

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

Build the Docker image from available binaries

Prior to building your image, follow these steps:

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

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

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

To build your image, follow these instructions:

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

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

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

```
docker images

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

Launch the Geometry service

There are methods for launching the Geometry service:

- You can use the PyAnsys Geometry launcher.
- You can manually launch the Geometry service.

Environment variables

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

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

You can also specify other optional environment variables:

- **ENABLE_TRACE**: Whether to set up the trace level for debugging purposes. The default is `0`, in which case the trace level is not set up. Options are `1` and `0`.
- **LOG_LEVEL**: Sets the Geometry service logging level. The default is `2`, in which case the logging level is `INFO`.

Here are some terms to keep in mind:

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

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

Using PyAnsys Geometry launcher

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

Linux/Mac

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

Powershell

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

Windows CMD

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

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

Manual launch

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

Geometry service launcher

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

Using PyAnsys Geometry launcher

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

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

modeler = launch_modeler()
```

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

Manual launch

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

Linux/Mac

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

Powershell

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

Windows CMD

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

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

Connect to the Geometry service

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

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

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

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

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

Environment variables

Linux/Mac

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

Powershell

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

Windows CMD

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

Modeler function

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

Go to Docker containers

Go to Getting started

1.5.2 Launch a local session

If Ansys 2023 R2 or later and PyAnsys Geometry are installed, you can create a local backend session using Discovery, SpaceClaim, or the Geometry service. Once the backend is running, PyAnsys Geometry can manage the connection.

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

Discovery

```
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()
```

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.

Go to Getting started

1.5.3 Launch a remote session

If a remote server is running Ansys 2023 R2 or later and is also running PIM (Product Instance Manager), you can use PIM to start a Discovery or SpaceClaim session that PyAnsys Geometry can connect to.

Warning: This option is only available for Ansys employees.

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

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")
```

(continues on next page)

(continued from previous page)

```
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.

End the session

To end the session, run the corresponding command:

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

Go to Getting started

1.5.4 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*.

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.

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
```


Go to Getting started

1.5.5 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.

Go to Getting started

1.5.6 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 or from the [PyAnsys Geometry repository](#) on GitHub.

Contents

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

Package dependencies

PyAnsys Geometry is supported on Python version 3.9 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

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
```

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
```

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 3.9 to 3.12. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

For example, on Linux with Python 3.9, unzip the wheelhouse archive and install it with these commands:

```
unzip ansys-geometry-core-v0.5.3-wheelhouse-Linux-3.9.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

If you are on Windows with Python 3.9, 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.

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

1.5.7 Frequently asked questions

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.

How is the Ansys Geometry Service installed?

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

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

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

However, when using the unified installer, it is necessary to pass in the `-geometryservice` flag to install it.

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

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

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

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

Where vXXX is the Ansys version that you have installed.

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.

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.

Go to Getting started

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.

2.1 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()*

2.1.1 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]))
```

2.2 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.

2.2.1 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-...
```

2.2.2 Direct API

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

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

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

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

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

2.3 Designer

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

2.3.1 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()
```

2.3.2 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)
```


2.3.3 Add materials to model

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

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

density = Quantity(125, 1000 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
poisson_ratio = Quantity(0.33, UNITS.dimensionless)
tensile_strength = Quantity(45)
material = Material(
    "steel",
    density,
    [MaterialProperty(MaterialPropertyType.POISSON_RATIO, "myPoisson", poisson_ratio)],
)
material.add_property(MaterialPropertyType.TENSILE_STRENGTH, "myTensile", Quantity(45))
design.add_material(material)
```

2.3.4 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))
```

2.3.5 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()*

2.3.6 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.sdocx"
design.download(file)
```

For more information, see the *Design* submodule.

2.4 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

2.5 Simple interactive example

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

2.5.1 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()
```

2.5.2 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])
```

(continues on next page)

(continued from previous page)

```
# 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="Content-...
```

API REFERENCE

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

3.1 The `ansys.geometry.core` library

3.1.1 Summary

Subpackages

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

Submodules

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

Attributes

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

Constants

<code>USE_TRAME</code>	Global constant for checking whether to use <code>trame</code>
<code>DISABLE_MULTIPLE_DESIGN_CHECK</code>	Global constant for disabling the <code>ensure_design_is_active</code> check.
<code>DOCUMENTATION_BUILD</code>	Global flag to set when building the documentation to use the proper PyVista Jupyter

The connection package

Summary

Submodules

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

The backend.py module

Summary

Enums

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

BackendType

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

Bases: `enum.Enum`

Provides an enum holding the available backend types.

Overview

Attributes

<code>DISCOVERY</code>
<code>SPACECLAIM</code>
<code>WINDOWS_SERVICE</code>
<code>LINUX_SERVICE</code>

Import detail

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

Attribute detail

`BackendType.DISCOVERY = 0`

`BackendType.SPACECLAIM = 1`

`BackendType.WINDOWS_SERVICE = 2`

`BackendType.LINUX_SERVICE = 3`

ApiVersions

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

Bases: `enum.Enum`

Provides an enum for all the compatibles API versions.

Overview

Attributes

<code>V_21</code>
<code>V_22</code>
<code>V_231</code>
<code>V_232</code>
<code>V_241</code>

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
```

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: beartype.typing.Optional[str] =
    DEFAULT_HOST, port:
    beartype.typing.Union[str, int] =
    DEFAULT_PORT, channel:
    beartype.typing.Optional[grpc.Channel] =
    None, remote_instance:
    beartype.typing.Optional[ansys.platform.instancemanagement.Instance] =
    None, docker_instance:
    beartype.typing.Optional[ansys.geometry.core.connection.docker.LocalDockerInstance] =
    None, product_instance:
    beartype.typing.Optional[ansys.geometry.core.connection.product.ProductInstance] =
    None, timeout:
    beartype.typing.Optional[ansys.geometry.core.typing.Real] =
    120, logging_level:
    beartype.typing.Optional[int] =
    logging.INFO, logging_file:
    beartype.typing.Optional[beartype.typing.Union[pathlib.Path,
    str]] = None, backend_type:
    beartype.typing.Optional[ansys.geometry.core.connection.backend.BackendType] =
    None)

```

Wraps the gRPC connection for the Geometry service.

Parameters

host

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

port

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

channel

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

remote_instance

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

docker_instance

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

product_instance

[ProductInstance, default: None] Corresponding local product instance when the product (Discovery or SpaceClaim) is launched through the launch_modeler_with_geometry_service(), launch_modeler_with_discovery() or the launch_modeler_with_spaceclaim() interface. This instance will be deleted when the GrpcClient.close method is called.

timeout

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

logging_level

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

logging_file

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

backend_type: BackendType, default: None

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

Overview**Methods**

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

Properties

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

Special methods

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

Import detail

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

Property detail

property GrpcClient.backend_type: *ansys.geometry.core.connection.backend.BackendType*

Backend type.

Options are Windows Service, Linux Service, Discovery, and SpaceClaim.

Notes

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

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

Get the current backend version.

Returns

Version

Backend version.

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

Flag indicating whether multiple designs are allowed.

Notes

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

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

Client gRPC channel.

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

Specific instance logger.

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

Flag indicating whether the client connection is closed.

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

Flag indicating whether the client channel is healthy.

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 PROTO API definition and stubs.

Module detail

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

Wait until a channel is healthy before returning.

Parameters

channel

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

timeout

[`float`] Timeout in seconds. An attempt is made every 100 milliseconds until the timeout is exceeded.

Raises

`TimeoutError`

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

The `conversions.py` module

Summary

Functions

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

Description

Module providing for conversions.

Module detail

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

Convert a `UnitVector3D` class to a unit vector Geometry service 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 Geometry service 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 Geometry service 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: beartype.typing.List[ansys.geometry.core.sketch.edge.SketchEdge], faces: beartype.typing.List[ansys.geometry.core.sketch.face.SketchFace], only_one_curve: beartype.typing.Optional[bool] = False) → ansys.api.geometry.v0.models_pb2.Geometries`

Convert lists of SketchEdge and SketchFace to a GRPCGeometries 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: beartype.typing.List[ansys.geometry.core.sketch.edge.SketchEdge], plane: ansys.geometry.core.math.plane.Plane) → beartype.typing.Tuple[beartype.typing.List[ansys.api.geometry.v0.models_pb2.Geometries], beartype.typing.List[ansys.api.geometry.v0.models_pb2.Arc]]`

Convert a list of SketchEdge to a GRPCGeometries 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 Geometry service 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 Geometry service 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 Geometry service 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 Geometry service gRPC message.

Parameters

point
 [Point3D] Source point data.

Returns

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

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

Convert a Point2D class to a point Geometry service 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 Geometry service 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 Geometry service gRPC message.

Parameters

segment
 [SketchSegment] Source segment data.

Returns

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

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

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

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

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

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

Convert an *ansys.api.geometry.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 an *ansys.api.geometry.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 an *ansys.api.geometry.CurveGeometry* 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* to an *ansys.api.geometry.CurveGeometry* 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 an `ansys.api.geometry.TrimmedCurve` gRPC message.

Parameters

curve
 [TrimmedCurve] Curve to convert.

Returns

GRPCTrimmedCurve
 Geometry service gRPC TrimmedCurve message.

The defaults.py module

Summary

Constants

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

Description

Module providing default connection parameters.

Module detail

`defaults.DEFAULT_HOST`

Default for the HOST name.

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

`defaults.DEFAULT_PORT`: **int**

Default for the HOST port.

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

`defaults.MAX_MESSAGE_LENGTH`

Default for the gRPC maximum message length.

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

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

Default for the Geometry service Docker image location.

Tag is dependent on what OS service is requested.

`defaults.DEFAULT_PIM_CONFIG`

Default for the PIM configuration when running PIM Light.

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

The `docker_instance.py` module

Summary

Classes

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

Enums

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

Functions

<i>get_geometry_container_type</i>	Given a <i>LocalDockerInstance</i> , provide back the <i>GeometryContainers</i> value.
------------------------------------	--

`LocalDockerInstance`

```
class ansys.geometry.core.connection.docker_instance.LocalDockerInstance(port: int =
    DEFAULT_PORT,
    connect_to_existing_service:
    bool = True,
    restart_if_existing_service:
    bool = False, name:
    beartype.typing.Optional[str]
    = None, image:
    beartype.typing.Optional[GeometryCon
    = 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

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

image

[Optional[`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.

Overview

Properties

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

Attributes

<code>__DOCKER_CLIENT__</code>	Docker client class variable. The default is <code>None</code> , in which case lazy
--------------------------------	---

Static methods

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

Import detail

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

Property detail

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

Docker container object that hosts the deployed Geometry service.

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

Flag indicating whether the container previously existed.

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

Attribute detail

LocalDockInstance.**__DOCKER_CLIENT__**: `docker.client.DockerClient`

Docker client class variable. The default is `None`, in which case lazy initialization is used.

Notes

`__DOCKER_CLIENT__` is a class variable, meaning that it is the same variable for all instances of this class.

Method detail

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

Get the initialized `__DOCKER_CLIENT__` object.

Returns

`DockerClient`

Initialized Docker client.

Notes

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

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

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

Returns

`bool`

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

GeometryContainers

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

Bases: `enum.Enum`

Provides an enum holding the available Geometry services.

Overview

Attributes

<code>WINDOWS_LATEST</code>
<code>LINUX_LATEST</code>
<code>WINDOWS_LATEST_UNSTABLE</code>
<code>LINUX_LATEST_UNSTABLE</code>
<code>WINDOWS_24_1</code>
<code>LINUX_24_1</code>
<code>WINDOWS_24_2</code>
<code>LINUX_24_2</code>

Import detail

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

Attribute detail

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

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

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

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

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

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

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

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

Description

Module for connecting to a local Docker container with the Geometry service.

Module detail

`docker_instance.get_geometry_container_type(instance: LocalDockerInstance) → beartype.typing.Union[GeometryContainers, None]`

Given a LocalDockerInstance, provide back the GeometryContainers value.

Parameters

instance

[LocalDockerInstance] The LocalDockerInstance object.

Returns

Union[GeometryContainers, None]

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

Notes

This method returns the first hit on the available tags.

The launcher.py module

Summary

Functions

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

Description

Module for connecting to instances of the Geometry service.

Module detail

`launcher.launch_modeler(mode: str = None, **kwargs: beartype.typing.Optional[beartype.typing.Dict]) → 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 `launch_remote_modeler()` and `launch_docker_modeler()` methods. Some of these keywords might be unused.

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(version: beartype.typing.Optional[str] = None, **kwargs: beartype.typing.Optional[beartype.typing.Dict]) → ansys.geometry.core.modeler.Modeler`

Start the Geometry service remotely using the PIM API.

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

Parameters

version

[str, default: None] Version of the Geometry service to run in the three-digit format. For example, "232". If you do not specify the version, the server chooses the version.

****kwargs**

[dict, default: None] Keyword arguments for the launching methods. For allowable keyword arguments, see the `launch_remote_modeler()` and `launch_docker_modeler()` methods. Some of these keywords might be unused.

Returns

`ansys.geometry.core.modeler.Modeler`

Instance of the Geometry service.

`launcher.launch_docker_modeler(port: int = DEFAULT_PORT, connect_to_existing_service: bool = True, restart_if_existing_service: bool = False, name: beartype.typing.Optional[str] = None, image: beartype.typing.Optional[ansys.geometry.core.connection.docker_instance.GeometryContainer] = None, **kwargs: beartype.typing.Optional[beartype.typing.Dict]) → ansys.geometry.core.modeler.Modeler`

Start the Geometry service locally using the `LocalDockerInstance` class.

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

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

image

[Optional[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.

****kwargs**

[dict, default: None] Keyword arguments for the launching methods. For allowable keyword arguments, see the `launch_remote_modeler()` and `launch_docker_modeler()` methods. Some of these keywords might be unused.

Returns

`Modeler`

Instance of the Geometry service.

`launcher.launch_modeler_with_discovery_and_pimlight(version: beartype.typing.Optional[str] = None) → ansys.geometry.core.modeler.Modeler`

Start Ansys Discovery remotely using the PIM API.

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

Parameters

version

[*str*, default: *None*] Version of Discovery to run in the three-digit format. For example, “232”. If you do not specify the version, the server chooses the version.

Returns

ansys.geometry.core.modeler.Modeler

Instance of Modeler.

`launcher.launch_modeler_with_geometry_service_and_pimlight`(*version: beartype.typing.Optional[str] = None*) → *ansys.geometry.core.modeler.Modeler*

Start the Geometry service remotely using the PIM API.

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

Parameters

version

[*str*, default: *None*] Version of the Geometry service to run in the three-digit format. For example, “232”. If you do not specify the version, the server chooses the version.

Returns

ansys.geometry.core.modeler.Modeler

Instance of Modeler.

`launcher.launch_modeler_with_spaceclaim_and_pimlight`(*version: beartype.typing.Optional[str] = None*) → *ansys.geometry.core.modeler.Modeler*

Start Ansys SpaceClaim remotely using the PIM API.

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

Parameters

version

[*str*, default: *None*] Version of SpaceClaim to run in the three-digit format. For example, “232”. If you do not specify the version, the server chooses the version.

Returns

ansys.geometry.core.modeler.Modeler

Instance of Modeler.

`launcher.launch_modeler_with_geometry_service`(*host: str = 'localhost', port: int = None, enable_trace: bool = False, log_level: int = 2, timeout: int = 60, logs_folder: str = 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**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.

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).

timeout

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

logs_folder

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

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

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

SystemError

If there is not an Ansys product 23.2 version or later installed a SystemError will be raised.

Examples

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

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

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

```
>>> from ansys.geometry.core import launch_modeler_with_geometry_service
>>> modeler = launch_modeler_with_geometry_service(host="10.171.22.44",
port=5001,
log_level=0,
```

(continues on next page)

(continued from previous page)

```
enable_trace= True,
timeout=300)
```

```
launcher.launch_modeler_with_discovery(product_version: int = None, host: str = 'localhost', port: int =
None, log_level: int = 2, api_version:
ansys.geometry.core.connection.backend.ApiVersions =
ApiVersions.LATEST, timeout: int = 150, manifest_path: str =
None, logs_folder: str = None, hidden: bool = False)
```

Start Ansys Discovery locally using the ProductInstance class.

Note: Support for Ansys Discovery is restricted to Ansys 24.1 onward.

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

Parameters

product_version: int, optional

The product version to be started. Goes from v23.2.1 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

Ansys products versions and their corresponding int values:

- 241 : Ansys 24R1

host: str, optional

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

port

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

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).

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.

logs_folder

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

hidden

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

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

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

SystemError:

If there is not an Ansys product 23.2 version or later installed or if a specific product's version is requested but not installed locally then a SystemError will be raised.

Examples

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

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

Starting an Ansys Discovery V 23.2 session, on address 10.171.22.44, port 5001, with chatty logs, using API v231 and a 300 seconds timeout:

```
>>> from ansys.geometry.core import launch_modeler_with_discovery
>>> modeler = launch_modeler_with_discovery(product_version = 232,
host="10.171.22.44",
port=5001,
log_level=0,
api_version= 231,
timeout=300)
```

```
launcher.launch_modeler_with_spaceclaim(product_version: int = None, host: str = 'localhost', port: int =
None, log_level: int = 2, api_version:
ansys.geometry.core.connection.backend.ApiVersions =
ApiVersions.LATEST, timeout: int = 150, manifest_path: str =
None, logs_folder: str = None, hidden: bool = False)
```

Start Ansys SpaceClaim locally using the ProductInstance class.

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

Parameters**product_version: int, optional**

The product version to be started. Goes from v23.2.1 to the latest. Default is None. If a specific product version is requested but not installed locally, a SystemError will be raised.

Ansys products versions and their corresponding int values:

- 232 : Ansys 23R2 SP1

- 241 : Ansys 24R1

host: str, optional

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

port

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

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).

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.

logs_folder

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

hidden

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

Returns

Modeler

Instance of the Geometry service.

Raises

ConnectionError

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

SystemError

If there is not an Ansys product 23.2 version or later installed or if a specific product's version is requested but not installed locally then a `SystemError` will be raised.

Examples

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

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

Starting an Ansys SpaceClaim V 23.2 session, on address 10.171.22.44, port 5001, with chatty logs, using API v231 and a 300 seconds timeout:

```
>>> from ansys.geometry.core import launch_modeler_with_spaceclaim
>>> modeler = launch_modeler_with_spaceclaim(product_version = 232,
      host="10.171.22.44",
      port=5001,
      log_level=0,
      api_version= 231,
      timeout=300)
```

The product_instance.py module

Summary

Classes

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

Functions

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

Constants

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

ProductInstance

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

ProductInstance class.

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

Parameters

pid

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

Overview

Methods

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

Import detail

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

Method detail

`ProductInstance.close()` → `bool`

Close the process associated to the pid.

Description

Module containing the `ProductInstance` class.

Module detail

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

Start the requested service locally using the `ProductInstance` class.

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

Parameters

product_version: ``int``, optional

The product version to be started. Goes from v23.2.1 to the latest. Default is `None`. If a specific product version is requested but not installed locally, a `SystemError` will be raised.

host: str, optional

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

port

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

enable_trace

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

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).

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.

logs_folder

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

hidden

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

Returns

Modeler

Instance of the Geometry service.

Raises

ConnectionError

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

SystemError

If there is not an Ansys product 23.2 version or later installed or if a specific product's version is requested but not installed locally then a `SystemError` will be raised.

`product_instance.get_available_port() → int`

Return an available port to be used.

Returns

`int`

The available port.

`product_instance.WINDOWS_GEOMETRY_SERVICE_FOLDER = 'GeometryService'`

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

`product_instance.DISCOVERY_FOLDER = 'Discovery'`

Default Discovery's folder name into the unified installer.

`product_instance.SPACECLAIM_FOLDER = 'scdm'`

Default SpaceClaim's folder name into the unified installer.

`product_instance.ADDINS_SUBFOLDER = 'Addins'`

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

`product_instance.BACKEND_SUBFOLDER = 'ApiServer'`

Default backend's folder name into the `ADDINS_SUBFOLDER` folder.

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

Default backend's addin filename.

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

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

The Windows Geometry Service's filename.

`product_instance.DISCOVERY_EXE = 'Discovery.exe'`

The Ansys Discovery's filename.

`product_instance.SPACECLAIM_EXE = 'SpaceClaim.exe'`

The Ansys SpaceClaim's filename.

`product_instance.BACKEND_LOG_LEVEL_VARIABLE = 'LOG_LEVEL'`

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

`product_instance.BACKEND_TRACE_VARIABLE = 'ENABLE_TRACE'`

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

`product_instance.BACKEND_HOST_VARIABLE = 'API_ADDRESS'`

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

`product_instance.BACKEND_PORT_VARIABLE = 'API_PORT'`

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

`product_instance.BACKEND_LOGS_FOLDER_VARIABLE = 'ANS_DSCO_REMOTE_LOGS_FOLDER'`

The backend's logs folder path to be used.

`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 addin 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 addin manifest file's path.

To be used only with Ansys Discovery and Ansys SpaceClaim.

The validate.py module

Summary

Functions

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

Description

Module to perform a connection validation check.

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

This command shows how this method is typically used:

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

Module detail

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

Create a client using the default settings and validate it.

Description

PyAnsys Geometry connection subpackage.

The designer package

Summary

Submodules

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

The beam.py module

Summary

Classes

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

BeamProfile

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

Represents a single beam profile organized within the design assembly.

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

Parameters

id

[str] Server-defined ID for the beam profile.

name

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

Notes

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

Overview

Properties

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

Import detail

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

Property detail

property BeamProfile.id: **str**

ID of the beam profile.

property BeamProfile.name: **str**

Name of the beam profile.

BeamCircularProfile

class ansys.geometry.core.designer.beam.BeamCircularProfile(*id: str, name: str, radius: 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 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.

radius

[Distance] Radius of the circle.

center: Point3D

3D point representing the center of the circle.

direction_x: UnitVector3D

X-axis direction.

direction_y: UnitVector3D

Y-axis direction.

Notes

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

Overview

Properties

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

Special methods

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

Import detail

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

Property detail

property BeamCircularProfile.*radius*: *ansys.geometry.core.misc.measurements.Distance*
Radius of the circular beam profile.

property BeamCircularProfile.*center*: *ansys.geometry.core.math.point.Point3D*
Center of the circular beam profile.

property BeamCircularProfile.*direction_x*: *ansys.geometry.core.math.vector.UnitVector3D*
X-axis direction of the circular beam profile.

property BeamCircularProfile.*direction_y*: *ansys.geometry.core.math.vector.UnitVector3D*
Y-axis direction of the circular beam profile.

Method detail

BeamCircularProfile.*__repr__*() → str
Represent the BeamCircularProfile as a string.

Beam

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

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

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

Parameters

- id**
[str] Server-defined ID for the body.
- name**
[str] User-defined label for the body.
- start**
[Point3D] Start of the beam line segment.
- end**
[Point3D] End of the beam line segment.
- profile**
[BeamProfile] Beam profile to use to create the beam.
- parent_component**
[Component] Parent component to nest the new beam under within the design assembly.

Overview

Properties

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

Special methods

<code>__repr__</code>	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:

beartype.typing.Union[*ansys.geometry.core.designer.component.Component*, **None**]

Component node that the beam is under.

property Beam.is_alive: **bool**

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

Method detail

Beam.__repr__() → str

Represent the beam as a string.

Description

Provides for creating and managing a beam.

The body.py module

Summary

Interfaces

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

Classes

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

Enums

<i>MidSurfaceOffsetType</i>	Provides values for mid-surface offsets supported by the Geometry service.
<i>CollisionType</i>	Provides values for collision types between bodies.

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

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

Methods

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

Import detail

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

Method detail

abstract `IBody.id()` → `str`

Get the ID of the body as a string.

abstract `IBody.name()` → `str`

Get the name of the body.

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

Get a list of all faces within the body.

Returns

`List[Face]`

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

Get a list of all edges within the body.

Returns

`List[Edge]`

abstract `IBody.is_alive()` → `bool`

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

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

Check if the body is a planar body.

abstract `IBody.surface_thickness()` → `beartype.typing.Union[pint.Quantity, None]`

Get the surface thickness of a surface body.

Notes

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

abstract `IBody.surface_offset()` → `beartype.typing.Union[MidSurfaceOffsetType, None]`

Get the surface offset type of a surface body.

Notes

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

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

Calculate the volume of the body.

Notes

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

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

Assign a material against the design in the active Geometry service instance.

Parameters

material

[Material] Source material data.

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

Add a mid-surface thickness to a surface body.

Parameters

thickness

[Quantity] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters

offset_type

[MidSurfaceOffsetType] Surface offset to assign.

Notes

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

abstract `IBody.imprint_curves(faces: beartype.typing.List[ansys.geometry.core.designer.face.Face], sketch: ansys.geometry.core.sketch.sketch.Sketch) → beartype.typing.Tuple[beartype.typing.List[ansys.geometry.core.designer.edge.Edge], beartype.typing.List[ansys.geometry.core.designer.face.Face]]`

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: List[Face]

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

sketch: Sketch

All curves to imprint on the faces.

Returns

Tuple[List[Edge], List[Face]]

All impacted edges and faces from the imprint operation.

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

Project all specified geometries onto the body.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

List[Face]

All faces from the project curves operation.

Notes

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

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

Project and imprint specified geometries onto the body.

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

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

List[Face]

All imprinted faces from the operation.

Notes

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

abstract `IBody.translate`(*direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *distance*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`) \rightarrow `None`

Translate the geometry body in the specified direction by a given distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: Union[~pint.Quantity, Distance, Real]

Distance (magnitude) of the translation.

Returns

`None`

abstract `IBody.rotate`(*axis_origin*: `ansys.geometry.core.math.point.Point3D`, *axis_direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *angle*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]`) \rightarrow `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: Union[~pint.Quantity, Angle, Real]

Angle (magnitude) of the rotation.

Returns

`None`

abstract `IBody.scale`(*value*: `ansys.geometry.core.typing.Real`) \rightarrow `None`

Scale the geometry body by the given value.

Parameters

value: Real

Value to scale the body by.

Notes

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

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

Map the geometry body to the new specified frame.

Parameters

frame: Frame

Structure defining the orientation of the body.

Notes

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

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

Mirror the geometry body across the specified plane.

Parameters

plane: Plane

Represents the mirror.

Notes

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

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

Get the collision state between bodies.

Parameters

body: Body

Object that the collision state is checked with.

Returns

CollisionType

Enum that defines the collision state between bodies.

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

Create a copy of the body and place it under the specified parent component.

Parameters

parent: Component

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

name: str

Name to give the new body.

Returns

Body

Copy of the body.

abstract `IBody.tessellate`(*merge*: *beartype.typing.Optional[bool] = False*) → *beartype.typing.Union[pyvista.PolyData, pyvista.MultiBlock]*

Tessellate the body and return the geometry as triangles.

Parameters

merge

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

Returns

PolyData, MultiBlock

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

Examples

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

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> my_comp = design.add_component("my-comp")
>>> body = my_comp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> blocks = body.tessellate()
>>> blocks
>>> MultiBlock (0x7f94ec757460)
  N Blocks: 6
  X Bounds: 0.000, 4.000
  Y Bounds: -1.000, 0.000
  Z Bounds: -0.500, 4.500
```

Merge the body:

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

abstract `IBody.plot`(*merge*: *bool = False*, *screenshot*: *beartype.typing.Optional[str] = None*, *use_frame*: *beartype.typing.Optional[bool] = None*, ***plotting_options*: *beartype.typing.Optional[dict]*) → `None`

Plot the body.

Parameters

merge

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

screenshot

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

use_frame

[bool, default: None] Whether to enable the use of `frame`. The default is None, in which case the USE_FRAME 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: beartype.typing.Union[Body, beartype.typing.Iterable[Body]]*) → None

Intersect two (or more) bodies.

Parameters

other

[Body] Body to intersect with.

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.

`IBody.subtract` (*other*: *beartype.typing.Union[Body, beartype.typing.Iterable[Body]]*) → `None`

Subtract two (or more) bodies.

Parameters

`other`

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

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.

`IBody.unite` (*other*: *beartype.typing.Union[Body, beartype.typing.Iterable[Body]]*) → `None`

Unite two (or more) bodies.

Parameters

`other`

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

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.

MasterBody

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

Bases: `IBody`

Represents solids and surfaces organized within the design assembly.

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

Parameters

`id`

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

`name`

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

`parent_component`

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

grpc_client

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

is_surface

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

Overview**Abstract methods**

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

Methods

<i>reset_tessellation_cache</i>	Decorate MasterBody methods that require a tessellation cache update.
<i>assign_material</i>	Assign a material against the design in the active Geometry service instance.
<i>add_midsurface_thickness</i>	Add a mid-surface thickness to a surface body.
<i>add_midsurface_offset</i>	Add a mid-surface offset to a surface body.
<i>translate</i>	Translate the geometry body in the specified direction by a given distance.
<i>rotate</i>	Rotate the geometry body around the specified axis by a given angle.
<i>scale</i>	Scale the geometry body by the given value.
<i>map</i>	Map the geometry body to the new specified frame.
<i>mirror</i>	Mirror the geometry body across the specified plane.
<i>get_collision</i>	Get the collision state between bodies.
<i>copy</i>	Create a copy of the body and place it under the specified parent component.
<i>tessellate</i>	Tessellate the body and return the geometry as triangles.

Properties

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

Special methods

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

Import detail

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

Property detail

property `MasterBody.id`: `str`

Get the ID of the body as a string.

property `MasterBody.name`: `str`

Get the name of the body.

property `MasterBody.is_surface`: `bool`

Check if the body is a planar body.

property `MasterBody.surface_thickness`: `beartype.typing.Union[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`: `beartype.typing.Union[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`: `beartype.typing.List[ansys.geometry.core.designer.face.Face]`

Get a list of all faces within the body.

Returns

`List[Face]`

property `MasterBody.edges`: `beartype.typing.List[ansys.geometry.core.designer.edge.Edge]`

Get a list of all edges within the body.

Returns

`List[Edge]`

property `MasterBody.is_alive`: `bool`

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

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

Calculate the volume of the body.

Notes

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

Method detail

MasterBody.**reset_tessellation_cache**()

Decorate MasterBody methods that require a 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 design in the active Geometry service instance.

Parameters

material

[Material] Source material data.

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

Add a mid-surface thickness to a surface body.

Parameters

thickness

[Quantity] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters

offset_type

[MidSurfaceOffsetType] Surface offset to assign.

Notes

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

abstract MasterBody.**imprint_curves**(*faces*: beartype.typing.List[ansys.geometry.core.designer.face.Face],
sketch: ansys.geometry.core.sketch.sketch.Sketch) →
 beartype.typing.Tuple[beartype.typing.List[ansys.geometry.core.designer.edge.Edge],
 beartype.typing.List[ansys.geometry.core.designer.face.Face]]

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: List[Face]

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

sketch: Sketch

All curves to imprint on the faces.

Returns

Tuple[List[Edge], List[Face]]

All impacted edges and faces from the imprint operation.

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

Project all specified geometries onto the body.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

List[Face]

All faces from the project curves operation.

Notes

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

abstract MasterBody.**imprint_projected_curves**(*direction*: ansys.geometry.core.math.vector.UnitVector3D, *sketch*: ansys.geometry.core.sketch.sketch.Sketch, *closest_face*: bool, *only_one_curve*: beartype.typing.Optional[bool] = False) → beartype.typing.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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`) → `None`

Translate the geometry body in the specified direction by a given distance.

Parameters**direction: UnitVector3D**

Direction of the translation.

distance: Union[~pint.Quantity, Distance, Real]

Distance (magnitude) of the translation.

Returns

`None`

`MasterBody.rotate`(*axis_origin*: `ansys.geometry.core.math.point.Point3D`, *axis_direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *angle*: `beartype.typing.Union[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: Union[~pint.Quantity, Angle, Real]

Angle (magnitude) of the rotation.

Returns

`None`

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

Scale the geometry body by the given value.

Parameters**value: Real**

Value to scale the body by.

Notes

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

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

Map the geometry body to the new specified frame.

Parameters

frame: Frame

Structure defining the orientation of the body.

Notes

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

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

Mirror the geometry body across the specified plane.

Parameters

plane: Plane

Represents the mirror.

Notes

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

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

Get the collision state between bodies.

Parameters

body: Body

Object that the collision state is checked with.

Returns

CollisionType

Enum that defines the collision state between bodies.

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

Create a copy of the body and place it under the specified parent component.

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*: *beartype.typing.Optional[bool]* = *False*, *transform*: *ansys.geometry.core.math.matrix.Matrix44* = *IDENTITY_MATRIX44*) → *beartype.typing.Union[pyvista.PolyData, pyvista.MultiBlock]*

Tessellate the body and return the geometry as triangles.

Parameters

merge

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

Returns

PolyData, MultiBlock

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

Examples

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

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> my_comp = design.add_component("my-comp")
>>> body = my_comp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> blocks = body.tessellate()
>>> blocks
>>> MultiBlock (0x7f94ec757460)
  N Blocks: 6
  X Bounds: 0.000, 4.000
  Y Bounds: -1.000, 0.000
  Z Bounds: -0.500, 4.500
```

Merge the body:

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

abstract `MasterBody.plot`(*merge*: *bool* = *False*, *screenshot*: *beartype.typing.Optional[str]* = *None*, *use_frame*: *beartype.typing.Optional[bool]* = *None*, ***plotting_options*: *beartype.typing.Optional[dict]*) → *None*

Plot the body.

Parameters

merge

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

screenshot

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

use_frame

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

**plotting_options

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

Examples

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

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

Plot the body and color each face individually:

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

abstract `MasterBody.intersect`(*other*: `beartype.typing.Union[Body, beartype.typing.Iterable[Body]]`) → `None`

Intersect two (or more) bodies.

Parameters

other

[`Body`] Body to intersect with.

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.

abstract `MasterBody.subtract`(*other: beartype.typing.Union[Body, beartype.typing.Iterable[Body]]*) → `None`

Subtract two (or more) bodies.

Parameters

other

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

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.

abstract `MasterBody.unite`(*other: beartype.typing.Union[Body, beartype.typing.Iterable[Body]]*) → `None`

Unite two (or more) bodies.

Parameters

other

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

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.

`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

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

Properties

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

Special methods

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

Import detail

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

Property detail

property `Body.id: str`

Get the ID of the body as a string.

property `Body.name: str`

Get the name of the body.

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

property `Body.faces: beartype.typing.List[ansys.geometry.core.designer.face.Face]`

Get a list of all faces within the body.

Returns

`List[Face]`

property `Body.edges: beartype.typing.List[ansys.geometry.core.designer.edge.Edge]`

Get a list of all edges within the body.

Returns

`List[Edge]`

property `Body.is_alive: bool`

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

property `Body.is_surface: bool`

Check if the body is a planar body.

property `Body.surface_thickness: beartype.typing.Union[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: beartype.typing.Union[MidSurfaceOffsetType, None]`

Get the surface offset type of a surface body.

Notes

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

property `Body.volume`: `pint.Quantity`

Calculate the volume of the body.

Notes

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

Method detail

`Body.reset_tessellation_cache()`

Decorate Body methods that require a tessellation cache update.

Parameters

`func`

[method] Method to call.

Returns

`Any`

Output of the method, if any.

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

Assign a material against the design in the active Geometry service instance.

Parameters

`material`

[Material] Source material data.

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

Add a mid-surface thickness to a surface body.

Parameters

`thickness`

[Quantity] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters

`offset_type`

[MidSurfaceOffsetType] Surface offset to assign.

Notes

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

Body.**imprint_curves**(*faces*: *beartype.typing.List*[*ansys.geometry.core.designer.face.Face*], *sketch*: *ansys.geometry.core.sketch.sketch.Sketch*) → *beartype.typing.Tuple*[*beartype.typing.List*[*ansys.geometry.core.designer.edge.Edge*], *beartype.typing.List*[*ansys.geometry.core.designer.face.Face*]]

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: List[Face]

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

sketch: Sketch

All curves to imprint on the faces.

Returns

Tuple[List[Edge], List[Face]]

All impacted edges and faces from the imprint operation.

Body.**project_curves**(*direction*: *ansys.geometry.core.math.vector.UnitVector3D*, *sketch*: *ansys.geometry.core.sketch.sketch.Sketch*, *closest_face*: *bool*, *only_one_curve*: *beartype.typing.Optional[bool] = False*) → *beartype.typing.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*: *beartype.typing.Optional[bool] = False*) → *beartype.typing.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.**translate**(*direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *distance*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`) → `None`

Translate the geometry body in the specified direction by a given distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: Union[~pint.Quantity, Distance, Real]

Distance (magnitude) of the translation.

Returns

None

Body.**rotate**(*axis_origin*: `ansys.geometry.core.math.point.Point3D`, *axis_direction*: `ansys.geometry.core.math.vector.UnitVector3D`, *angle*: `beartype.typing.Union[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: Union[~pint.Quantity, Angle, Real]

Angle (magnitude) of the rotation.

Returns**None**

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

Scale the geometry body by the given value.

Parameters**value: Real**

Value to scale the body by.

Notes

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

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

Map the geometry body to the new specified frame.

Parameters**frame: Frame**

Structure defining the orientation of the body.

Notes

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

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

Mirror the geometry body across the specified plane.

Parameters**plane: Plane**

Represents the mirror.

Notes

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

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

Get the collision state between bodies.

Parameters**body: Body**

Object that the collision state is checked with.

Returns**CollisionType**

Enum that defines the collision state between bodies.

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

Create a copy of the body and place it under the specified parent component.

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: beartype.typing.Optional[bool] = False*) → beartype.typing.Union[pyvista.PolyData, pyvista.MultiBlock]

Tessellate the body and return the geometry as triangles.

Parameters**merge**

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

Returns**PolyData, MultiBlock**

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

Examples

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

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 0, 1])
>>> sketch = Sketch(plane)
>>> box = sketch.box(Point2D([2, 0]), 4, 4)
>>> design = modeler.create_design("my-design")
>>> my_comp = design.add_component("my-comp")
>>> body = my_comp.extrude_sketch("my-sketch", sketch, 1 * u.m)
>>> blocks = body.tessellate()
>>> blocks
>>> MultiBlock (0x7f94ec757460)
  N Blocks: 6
  X Bounds: 0.000, 4.000
  Y Bounds: -1.000, 0.000
  Z Bounds: -0.500, 4.500
```

Merge the body:

```
>>> mesh = body.tessellate(merge=True)
>>> mesh
PolyData (0x7f94ec75f3a0)
  N Cells: 12
  N Points: 24
  X Bounds: 0.000e+00, 4.000e+00
```

(continues on next page)

(continued from previous page)

```

Y Bounds:      -1.0000e+00,  0.0000e+00
Z Bounds:      -5.0000e-01,  4.5000e+00
N Arrays:      0

```

`Body.plot`(merge: *bool* = *False*, screenshot: *beartype.typing.Optional[str]* = *None*, use_trame: *beartype.typing.Optional[bool]* = *None*, ***plotting_options*: *beartype.typing.Optional[dict]*) → *None*

Plot the body.

Parameters

merge

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

screenshot

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

use_trame

[*bool*, default: *None*] Whether to enable the use of *trame*. The default is *None*, in which case the *USE_TRAME* 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*: *beartype.typing.Union[Body, beartype.typing.Iterable[Body]]*) → *None*

Intersect two (or more) bodies.

Parameters

other

[*Body*] Body to intersect with.

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.

`Body.subtract(other: beartype.typing.Union[Body, beartype.typing.Iterable[Body]]) → None`

Subtract two (or more) bodies.

Parameters**other**

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

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.

`Body.unite(other: beartype.typing.Union[Body, beartype.typing.Iterable[Body]]) → None`

Unite two (or more) bodies.

Parameters**other**

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

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.

`Body.__repr__()` → `str`

Represent the `Body` as a string.

MidSurfaceOffsetType

`class ansys.geometry.core.designer.body.MidSurfaceOffsetType(*args, **kws)`

Bases: `enum.Enum`

Provides values for mid-surface offsets supported by the Geometry service.

Overview

Attributes

<i>MIDDLE</i>
<i>TOP</i>
<i>BOTTOM</i>
<i>VARIABLE</i>
<i>CUSTOM</i>

Import detail

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

Attribute detail

MidSurfaceOffsetType.**MIDDLE** = 0

MidSurfaceOffsetType.**TOP** = 1

MidSurfaceOffsetType.**BOTTOM** = 2

MidSurfaceOffsetType.**VARIABLE** = 3

MidSurfaceOffsetType.**CUSTOM** = 4

CollisionType

```
class ansys.geometry.core.designer.body.CollisionType(*args, **kws)
```

Bases: `enum.Enum`

Provides values for collision types between bodies.

Overview

Attributes

<i>NONE</i>
<i>TOUCH</i>
<i>INTERSECT</i>
<i>CONTAINED</i>
<i>CONTAINEDTOUCH</i>

Import detail

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

Attribute detail

`CollisionType.NONE` = 0

`CollisionType.TOUCH` = 1

`CollisionType.INTERSECT` = 2

`CollisionType.CONTAINED` = 3

`CollisionType.CONTAINEDTOUCH` = 4

Description

Provides for managing a body.

The `component.py` module

Summary

Classes

<code>Component</code>	Provides for creating and managing a component.
------------------------	---

Enums

<code>SharedTopologyType</code>	Enum for the component shared topologies available in the Geometry service.
<code>ExtrusionDirection</code>	Enum for extrusion direction definition.

Component

```
class ansys.geometry.core.designer.component.Component(name: str, parent_component:
    beartype.typing.Union[Component, None],
    grpc_client: an-
    sys.geometry.core.connection.client.GrpcClient,
    template:
    beartype.typing.Optional[Component] =
    None, preexisting_id:
    beartype.typing.Optional[str] = None,
    master_component:
    beartype.typing.Optional[ansys.geometry.core.designer.part.M
    = 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.

preexisting_id

[[str](#), default: [None](#)] ID of a component pre-existing on the server side to use to create the component on the client-side data model. If an ID is specified, a new component is not created on the server.

master_component

[[MasterComponent](#), default: [None](#)] Master component to use to create a nested component instance instead of creating a new component.

read_existing_comp

[[bool](#), default: [False](#)] Whether an existing component on the service should be read. This parameter is only valid when connecting to an existing service session. Otherwise, avoid using this optional parameter.

Overview

Methods

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

Properties

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

Special methods

```
__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.components: beartype.typing.List[Component]`

List of Component objects inside of the component.

property `Component.bodies: beartype.typing.List[ansys.geometry.core.designer.body.Body]`

List of Body objects inside of the component.

property `Component.beams: beartype.typing.List[ansys.geometry.core.designer.beam.Beam]`

List of Beam objects inside of the component.

property `Component.design_points:`
`beartype.typing.List[ansys.geometry.core.designer.designpoint.DesignPoint]`

List of DesignPoint objects inside of the component.

property `Component.coordinate_systems:`
`beartype.typing.List[ansys.geometry.core.designer.coordinate_system.CoordinateSystem]`

List of CoordinateSystem objects inside of the component.

property `Component.parent_component: beartype.typing.Union[Component, None]`

Parent of the component.

property `Component.is_alive: bool`

Whether the component is still alive on the server side.

property `Component.shared_topology: beartype.typing.Union[SharedTopologyType, None]`

Shared topology type of the component (if any).

Notes

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

Method detail

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

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

Returns

Matrix44

4x4 transformation matrix of the component in world space.

Component.**modify_placement**(*translation:*

beartype.typing.Optional[ansys.geometry.core.math.vector.Vector3D] = None,

rotation_origin:

beartype.typing.Optional[ansys.geometry.core.math.point.Point3D] = None,

rotation_direction:

beartype.typing.Optional[ansys.geometry.core.math.vector.UnitVector3D] =

None, rotation_angle: beartype.typing.Union[pint.Quantity,

ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]
= 0)

Apply a translation and/or rotation to the existing 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

[Union[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: beartype.typing.Optional[Component] = 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: beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real], direction: beartype.typing.Union[ExtrusionDirection, str] = ExtrusionDirection.POSITIVE) → ansys.geometry.core.designer.body.Body`

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

Parameters

name

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

sketch

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

distance

[Union[Quantity, Distance, Real]] Distance to extrude the solid body.

direction

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

Returns

Body

Extruded body from the given sketch.

Notes

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

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

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

Parameters

name

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

sketch

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

path

[List[TrimmedCurve]] The path to sweep the profile along.

Returns**Body**

Created body from the given sketch.

Notes

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

```
Component.sweep_chain(name: str, path:  
    beartype.typing.List[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve],  
    chain:  
    beartype.typing.List[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve])  
    → ansys.geometry.core.designer.body.Body
```

Create a body by sweeping a chain of curves along a path.

Parameters**name**

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

path

[List[TrimmedCurve]] The path to sweep the chain along.

chain

[List[TrimmedCurve]] A chain of trimmed curves.

Returns**Body**

Created body from the given sketch.

Notes

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

```
Component.extrude_face(name: str, face: ansys.geometry.core.designer.face.Face, distance:  
    beartype.typing.Union[pint.Quantity,  
    ansys.geometry.core.misc.measurements.Distance], direction:  
    beartype.typing.Union[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

[Union[Quantity, Distance, Real]] Distance to extrude the solid body.

direction

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

Returns**Body**

Extruded solid body.

Notes

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

Component.**create_sphere**(*name*: str, *center*: ansys.geometry.core.math.point.Point3D, *radius*: ansys.geometry.core.misc.measurements.Distance) → *ansys.geometry.core.designer.body.Body*

Create a sphere body defined by the center point and the radius.

Parameters**name**

[str] Body name.

center

[Point3D] Center point of the sphere.

radius

[Distance] Radius of the sphere.

Returns**Body**

Sphere body object.

Component.**create_body_from_loft_profile**(*name*: str, *profiles*: *beartype.typing.List[beartype.typing.List[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]]*, *periodic*: bool = False, *ruled*: bool = False) → *ansys.geometry.core.designer.body.Body*

Create a lofted body from a collection of trimmed curves.

Parameters**name**

[str] Name of the lofted body.

profiles

[List[List[TrimmedCurve]]] Collection of lists of trimmed curves (profiles) defining the lofted body's shape.

periodic

[bool, default: False] Whether the lofted body should have periodic continuity.

ruled

[bool] Whether the lofted body should be ruled.

Returns

Body

Created lofted body object.

Notes

Surfaces produced have a U parameter in the direction of the profile curves, and a V parameter in the direction of lofting. Profiles can have different numbers of segments. A minimum twist solution is produced. Profiles should be all closed or all open. Closed profiles cannot contain inner loops. If closed profiles are supplied, a closed (solid) body is produced, if possible. Otherwise, an open (sheet) body is produced. The periodic argument applies when the profiles are closed. It is ignored if the profiles are open.

If `periodic=True`, at least three profiles must be supplied. The loft continues from the last profile back to the first profile to produce surfaces that are periodic in V.

If `periodic=False`, at least two profiles must be supplied. If the first and last profiles are planar, end capping faces are created. Otherwise, an open (sheet) body is produced. If `ruled=True`, separate ruled surfaces are produced between each pair of profiles. If `periodic=True`, the loft continues from the last profile back to the first profile, but the surfaces are not periodic.

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

Create a surface body with a sketch profile.

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

Parameters**name**

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

sketch

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

Returns**Body**

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

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

Create a surface body based on a face.

Parameters**name**

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

face

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

Returns**Body**

Surface body.

Notes

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

Component.**create_coordinate_system**(*name*: *str*, *frame*: *ansys.geometry.core.math.frame.Frame*) → *ansys.geometry.core.designer.coordinate_system.CoordinateSystem*

Create a coordinate system.

The newly created coordinate system is place under this component within the design assembly.

Parameters

name

[*str*] User-defined label for the new coordinate system.

frame

[*Frame*] Frame defining the coordinate system bounds.

Returns

CoordinateSystem

Component.**translate_bodies**(*bodies*: *beartype.typing.List[ansys.geometry.core.designer.body.Body]*, *direction*: *ansys.geometry.core.math.vector.UnitVector3D*, *distance*: *beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]*) → *None*

Translate the geometry bodies in a specified direction by a given distance.

Parameters

bodies: List[Body]

List of bodies to translate by the same distance.

direction: UnitVector3D

Direction of the translation.

distance: Union[~pint.Quantity, Distance, Real]

Magnitude of the translation.

Returns

None

Notes

If the body does not belong to this component (or its children), it is not translated.

Component.**create_beams**(*segments*: *beartype.typing.List[beartype.typing.Tuple[ansys.geometry.core.math.point.Point3D, ansys.geometry.core.math.point.Point3D]]*, *profile*: *ansys.geometry.core.designer.beam.BeamProfile*) → *beartype.typing.List[ansys.geometry.core.designer.beam.Beam]*

Create beams under the component.

Parameters

segments

[*List[Tuple[Point3D, Point3D]]*] List of start and end pairs, each specifying a single line segment.

profile

[BeamProfile] Beam profile to use to create the beams.

Notes

The newly created beams synchronize to a design within a supporting Geometry service instance.

Component.**create_beam**(*start*: ansys.geometry.core.math.point.Point3D, *end*: ansys.geometry.core.math.point.Point3D, *profile*: ansys.geometry.core.designer.beam.BeamProfile) → *ansys.geometry.core.designer.beam.Beam*

Create a beam under the component.

The newly created beam synchronizes to a design within a supporting Geometry service instance.

Parameters

start

[Point3D] Starting point of the beam line segment.

end

[Point3D] Ending point of the beam line segment.

profile

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

Component.**delete_component**(*component*: beartype.typing.Union[Component, str]) → None

Delete a component (itself or its children).

Parameters

component

[Union[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*: beartype.typing.Union[ansys.geometry.core.designer.body.Body, str]) → None

Delete a body belonging to this component (or its children).

Parameters

body

[Union[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: beartype.typing.List[ansys.geometry.core.math.point.Point3D]) → beartype.typing.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: beartype.typing.Union[ansys.geometry.core.designer.beam.Beam, str]) → None`
Delete an existing beam belonging to this component (or its children).

Parameters**beam**

[Union[Beam, str]] ID of the beam or instance to delete.

Notes

If the beam does not belong to this component (or its children), it is not deleted.

`Component.search_component(id: str) → beartype.typing.Union[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) → beartype.typing.Union[ansys.geometry.core.designer.body.Body, None]`
Search bodies in the component and nested components recursively for a body.

Parameters**id**

[str] ID of the body to search for.

Returns**Body**

Body with the requested ID. If the ID is not found, None is returned.

`Component.search_beam(id: str) → beartype.typing.Union[ansys.geometry.core.designer.beam.Beam, None]`
Search beams in the component and nested components recursively for a beam.

Parameters**id**

[str] ID of the beam to search for.

Returns**Union[Beam, None]**

Beam with the requested ID. If the ID is not found, None is returned.

`Component.tessellate(merge_component: bool = False, merge_bodies: bool = False) →`
`beartype.typing.Union[pyvista.PolyData, pyvista.MultiBlock]`

Tessellate the component.

Parameters**merge_component**

[bool, default: False] Whether to merge this component into a single dataset. When True, all the individual bodies are effectively combined into a single dataset without any hierarchy.

merge_bodies

[bool, default: False] Whether to merge each body into a single dataset. When True, all the faces of each individual body are effectively merged into a single dataset without separating faces.

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

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

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

Component `.plot` (*merge_component: bool = False, merge_bodies: bool = False, screenshot: beartype.typing.Optional[str] = None, use_trame: beartype.typing.Optional[bool] = None, **plotting_options: beartype.typing.Optional[dict]*) → None

Plot the component.

Parameters

merge_component

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

merge_bodies

[bool, default: False] Whether to merge each body into a single dataset. When True, all the faces of each individual body are effectively merged into a single dataset without separating faces.

screenshot

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

use_trame

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

****plotting_options**

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

Examples

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

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

(continues on next page)

(continued from previous page)

```
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.

SharedTopologyType

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

Bases: `enum.Enum`

Enum for the component shared topologies available in the Geometry service.

Overview

Attributes

<code>SHARETYPE_NONE</code>
<code>SHARETYPE_SHARE</code>
<code>SHARETYPE_MERGE</code>
<code>SHARETYPE_GROUPS</code>

Import detail

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

Attribute detail

`SharedTopologyType.SHARETYPE_NONE = 0`

`SharedTopologyType.SHARETYPE_SHARE = 1`

`SharedTopologyType.SHARETYPE_MERGE = 2`

`SharedTopologyType.SHARETYPE_GROUPS = 3`

ExtrusionDirection

class ansys.geometry.core.designer.component.**ExtrusionDirection**(*args, **kwds)

Bases: enum.Enum

Enum for extrusion direction definition.

Overview

Constructors

<i>from_string</i>	Convert a string to an ExtrusionDirection enum.
--------------------	---

Attributes

<i>POSITIVE</i>
<i>NEGATIVE</i>

Import detail

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

Attribute detail

ExtrusionDirection.**POSITIVE** = '+'

ExtrusionDirection.**NEGATIVE** = '-'

Method detail

classmethod ExtrusionDirection.**from_string**(string: str, use_default_if_error: bool = False) → ExtrusionDirection

Convert a string to an ExtrusionDirection enum.

Description

Provides for managing components.

The coordinate_system.py module

Summary

Classes

<i>CoordinateSystem</i>	Represents a user-defined coordinate system within the design assembly.
-------------------------	---

CoordinateSystem

```
class ansys.geometry.core.designer.coordinate_system.CoordinateSystem(name: str, frame: an-
sys.geometry.core.math.frame.Frame,
parent_component: an-
sys.geometry.core.designer.component.Com
grpc_client: an-
sys.geometry.core.connection.client.GrpcCl
preexisting_id:
beartype.typing.Optional[str]
= None)
```

Represents a user-defined coordinate system within the design assembly.

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

Parameters

name

[str] User-defined label for the coordinate system.

frame

[Frame] Frame defining the coordinate system bounds.

parent_component

[Component, default: Component] Parent component the coordinate system is assigned against.

grpc_client

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

Overview

Properties

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

Special methods

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

Import detail

```
from ansys.geometry.core.designer.coordinate_system import CoordinateSystem
```

Property detail

property `CoordinateSystem.id: str`

ID of the coordinate system.

property `CoordinateSystem.name: str`

Name of the coordinate system.

property `CoordinateSystem.frame: ansys.geometry.core.math.frame.Frame`

Frame of the coordinate system.

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

Parent component of the coordinate system.

property `CoordinateSystem.is_alive: bool`

Flag indicating if coordinate system is still alive on the server side.

Method detail

`CoordinateSystem.__repr__()` → `str`

Represent the coordinate system as a string.

Description

Provides for managing a user-defined coordinate system.

The `design.py` module

Summary

Classes

<i>Design</i>	Provides for organizing geometry assemblies.
---------------	--

Enums

<i>DesignFileFormat</i>	Provides supported file formats that can be downloaded for designs.
-------------------------	---

Design

```
class ansys.geometry.core.designer.design.Design(name: str, modeler:
                                             ansys.geometry.core.modeler.Modeler,
                                             read_existing_design: bool = False)
```

Bases: *ansys.geometry.core.designer.component.Component*

Provides for organizing geometry assemblies.

This class synchronizes to a supporting Geometry service instance.

Parameters

name

[str] User-defined label for the design.

grpc_client

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

read_existing_design

[bool, default: False] Whether an existing design on the service should be read. This parameter is only valid when connecting to an existing service session. Otherwise, avoid using this optional parameter.

Overview

Methods

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

Properties

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

Special methods

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

Import detail

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

Property detail

property `Design.design_id: str`

The design's object unique id.

property `Design.materials:`

beartype.typing.List`[ansys.geometry.core.materials.material.Material]`

List of materials available for the design.

property `Design.named_selections:`

beartype.typing.List`[ansys.geometry.core.designer.selection.NamedSelection]`

List of named selections available for the design.

property `Design.beam_profiles:`

beartype.typing.List`[ansys.geometry.core.designer.beam.BeamProfile]`

List of beam profile available for the design.

property `Design.is_active: bool`

Whether the design is currently active.

Method detail

`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: beartype.typing.Union[pathlib.Path, str]) → None`

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

Parameters

file_location

[Union[Path, str]] Location on disk to save the file to.

`Design.download(file_location: beartype.typing.Union[pathlib.Path, str], format: beartype.typing.Optional[DesignFileFormat] = DesignFileFormat.SCDOCX) → None`

Export and download the design from the active Geometry server instance.

Parameters

file_location

[Union[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: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to an scdocx file.

Parameters

location

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

Returns

str

The path to the saved file.

`Design.export_to_parasolid_text(location: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to a Parasolid text file.

Parameters

location

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

Returns

str

The path to the saved file.

`Design.export_to_parasolid_bin(location: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to a Parasolid binary file.

Parameters

location

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

Returns

str

The path to the saved file.

`Design.export_to_fmd(location: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to an FMD file.

Parameters

location

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

Returns

`str`

The path to the saved file.

`Design.export_to_step(location: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to a STEP file.

Parameters

location

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

Returns

`str`

The path to the saved file.

`Design.export_to_iges(location: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to an IGES file.

Parameters

location

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

Returns

`str`

The path to the saved file.

`Design.export_to_pmdb(location: beartype.typing.Union[pathlib.Path, str] = None) → str`

Export the design to a PMDB file.

Parameters

location

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

Returns

`str`

The path to the saved file.

```
Design.create_named_selection(name: str, bodies:
    beartype.typing.Optional[beartype.typing.List[ansys.geometry.core.designer.body.Body]]
    = None, faces:
    beartype.typing.Optional[beartype.typing.List[ansys.geometry.core.designer.face.Face]]
    = None, edges:
    beartype.typing.Optional[beartype.typing.List[ansys.geometry.core.designer.edge.Edge]]
    = None, beams:
    beartype.typing.Optional[beartype.typing.List[ansys.geometry.core.designer.beam.Beam]]
    = None, design_points:
    beartype.typing.Optional[beartype.typing.List[ansys.geometry.core.designer.designpoint.Designpoint]]
    = None) → ansys.geometry.core.designer.selection.NamedSelection
```

Create a named selection on the active Geometry server instance.

Parameters

name

[str] User-defined name for the named selection.

bodies

[List[Body], default: None] All bodies to include in the named selection.

faces

[List[Face], default: None] All faces to include in the named selection.

edges

[List[Edge], default: None] All edges to include in the named selection.

beams

[List[Beam], default: None] All beams to include in the named selection.

design_points

[List[DesignPoint], default: None] All design points to include in the named selection.

Returns

NamedSelection

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

```
Design.delete_named_selection(named_selection:
    beartype.typing.Union[ansys.geometry.core.designer.selection.NamedSelection,
    str]) → None
```

Delete a named selection on the active Geometry server instance.

Parameters

named_selection

[Union[NamedSelection, str]] Name of the named selection or instance.

```
Design.delete_component(component:
    beartype.typing.Union[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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance]`, *center*: `beartype.typing.Union[numpy.ndarray, ansys.geometry.core.typing.RealSequence, ansys.geometry.core.math.point.Point3D] = ZERO_POINT3D`, *direction_x*: `beartype.typing.Union[numpy.ndarray, ansys.geometry.core.typing.RealSequence, ansys.geometry.core.math.vector.UnitVector3D, ansys.geometry.core.math.vector.Vector3D] = UNITVECTOR3D_X`, *direction_y*: `beartype.typing.Union[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 the creating beams.

Parameters

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

radius
[`Real`] Radius of the beam circular profile.

center
[`Union[ndarray, RealSequence, Point3D]`] Center of the beam circular profile.

direction_x
[`Union[ndarray, RealSequence, UnitVector3D, Vector3D]`] X-plane direction.

direction_y
[`Union[ndarray, RealSequence, UnitVector3D, Vector3D]`] Y-plane direction.

`Design.add_midsurface_thickness`(*thickness*: `pint.Quantity`, *bodies*: `beartype.typing.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*: *beartype.typing.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*: *beartype.typing.Union*[ansys.geometry.core.designer.beam.BeamProfile, *str*]) → None

Remove a beam profile on the active geometry server instance.

Parameters

beam_profile

[Union[BeamProfile, *str*]] A beam profile name or instance that should be deleted.

`Design.insert_file`(*file_location*: *beartype.typing.Union*[*pathlib.Path*, *str*]) → *ansys.geometry.core.designer.component.Component*

Insert a file into the design.

Parameters

file_location

[Union[Path, *str*]] Location on disk where the file is located.

Returns

Component

The newly inserted component.

`Design.__repr__`() → *str*

Represent the Design as a string.

DesignFileFormat

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

Bases: `enum.Enum`

Provides supported file formats that can be downloaded for designs.

Overview

Attributes

<i>SCDOCX</i>
<i>PARASOLID_TEXT</i>
<i>PARASOLID_BIN</i>
<i>FMD</i>
<i>STEP</i>
<i>IGES</i>
<i>PMDB</i>
<i>INVALID</i>

Import detail

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

Attribute detail

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

```
DesignFileFormat.PARASOLID_TEXT = ('PARASOLID_TEXT',)
```

```
DesignFileFormat.PARASOLID_BIN = ('PARASOLID_BIN',)
```

```
DesignFileFormat.FMD = ('FMD',)
```

```
DesignFileFormat.STEP = ('STEP',)
```

```
DesignFileFormat.IGES = ('IGES',)
```

```
DesignFileFormat.PMDB = ('PMDB',)
```

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

Description

Provides for managing designs.

The designpoint.py module

Summary

Classes

<i>DesignPoint</i>	Provides for creating design points in components.
--------------------	--

DesignPoint

```
class ansys.geometry.core.designer.designpoint.DesignPoint(id: str, name: str, point: ansys.geometry.core.math.point.Point3D, parent_component: ansys.geometry.core.designer.component.Component)
```

Provides for creating design points in components.

Parameters

id

[*str*] Server-defined ID for the design points.

name

[*str*] User-defined label for the design points.

points

[*Point3D*] 3D point constituting the design points.

parent_component

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

Overview

Properties

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

Special methods

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

Import detail

```
from ansys.geometry.core.designer.designpoint import DesignPoint
```

Property detail

property `DesignPoint.id: str`

ID of the design point.

property `DesignPoint.name: str`

Name of the design point.

property `DesignPoint.value: ansys.geometry.core.math.point.Point3D`

Value of the design point.

property `DesignPoint.parent_component:`

beartype.typing.Union`[ansys.geometry.core.designer.component.Component, None]`

Component node that the design point is under.

Method detail

`DesignPoint.__repr__()` → `str`

Represent the design points as a string.

Description

Module for creating and managing design points.

The `edge.py` module

Summary

Classes

<i>Edge</i>	Represents a single edge of a body within the design assembly.
-------------	--

Enums

CurveType	Provides values for the curve types supported by the Geometry service.
------------------	--

Edge

```
class ansys.geometry.core.designer.edge.Edge(id: str, curve_type: CurveType, body:
    ansys.geometry.core.designer.body.Body, grpc_client:
    ansys.geometry.core.connection.client.GrpcClient,
    is_reversed: bool = False)
```

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

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

Parameters

id

[str] Server-defined ID for the body.

curve_type

[CurveType] Type of curve that the edge forms.

body

[Body] Parent body that the edge constructs.

grpc_client

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

is_reversed

[bool] Direction of the edge.

Overview

Properties

<i>id</i>	ID of the edge.
<i>is_reversed</i>	Flag indicating if the edge is reversed.
<i>shape</i>	Underlying trimmed curve of the edge.
<i>length</i>	Calculated length of the edge.
<i>curve_type</i>	Curve type of the edge.
<i>faces</i>	Faces that contain the edge.
<i>start</i>	Start point of the edge.
<i>end</i>	End point of the edge.

Import detail

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

Property detail

property Edge.**id**: **str**

ID of the edge.

property Edge.**is_reversed**: **bool**

Flag indicating if the edge is reversed.

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

Underlying trimmed curve of the edge.

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

property Edge.**length**: **pint.Quantity**

Calculated length of the edge.

property Edge.**curve_type**: **CurveType**

Curve type of the edge.

property Edge.**faces**: **beartype.typing.List[ansys.geometry.core.designer.face.Face]**

Faces that contain the edge.

property Edge.**start**: **ansys.geometry.core.math.point.Point3D**

Start point of the edge.

property Edge.**end**: **ansys.geometry.core.math.point.Point3D**

End point of the edge.

CurveType

class ansys.geometry.core.designer.edge.**CurveType**(*args, **kwargs)

Bases: **enum.Enum**

Provides values for the curve types supported by the Geometry service.

Overview

Attributes

<i>CURVETYPE_UNKNOWN</i>
<i>CURVETYPE_LINE</i>
<i>CURVETYPE_CIRCLE</i>
<i>CURVETYPE_ELLIPSE</i>
<i>CURVETYPE_NURBS</i>
<i>CURVETYPE_PROCEDURAL</i>

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

<i>FaceLoop</i>	Provides an internal class holding the face loops defined on the server side.
<i>Face</i>	Represents a single face of a body within the design assembly.

Enums

<i>SurfaceType</i>	Provides values for the surface types supported by the Geometry service.
<i>FaceLoopType</i>	Provides values for the face loop types supported by the Geometry service.

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:
                                                beartype.typing.List[ansys.geometry.core.designer.edge.Edge])
```

Provides an internal class holding the face loops defined on the server side.

Parameters

type	[FaceLoopType] Type of loop.
length	[Quantity] Length of the loop.
min_bbox	[Point3D] Minimum point of the bounding box containing the loop.
max_bbox	[Point3D] Maximum point of the bounding box containing the loop.
edges	[List[Edge]] Edges contained in the loop.

Notes

This class is to be used only when parsing server side results. It is not intended to be instantiated by a user.

Overview

Properties

<i>type</i>	Type of the loop.
<i>length</i>	Length of the loop.
<i>min_bbox</i>	Minimum point of the bounding box containing the loop.
<i>max_bbox</i>	Maximum point of the bounding box containing the loop.
<i>edges</i>	Edges contained in the loop.

Import detail

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

Property detail

property FaceLoop.type: *FaceLoopType*

Type of the loop.

property FaceLoop.length: *pint.Quantity*

Length of the loop.

property FaceLoop.min_bbox: *ansys.geometry.core.math.point.Point3D*

Minimum point of the bounding box containing the loop.

property FaceLoop.max_bbox: *ansys.geometry.core.math.point.Point3D*

Maximum point of the bounding box containing the loop.

property FaceLoop.edges: *beartype.typing.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

<i>normal</i>	Get the normal direction to the face at certain proportional UV coordinates.
<i>point</i>	Get a point of the face evaluated at certain proportional UV coordinates.
<i>create_isoparametric_curves</i>	Create isoparametric curves at the given proportional parameter.

Properties

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

Import detail

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

Property detail

property `Face.id`: `str`

Face ID.

property `Face.is_reversed`: `bool`

Flag indicating if the face is reversed.

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

Body that the face belongs to.

property `Face.shape`: `ansys.geometry.core.shapes-surfaces-trimmed-surface.TrimmedSurface`

Underlying trimmed surface of the face.

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

property `Face.surface_type`: `SurfaceType`

Surface type of the face.

property `Face.area`: `pint.Quantity`

Calculated area of the face.

property `Face.edges`: `beartype.typing.List[ansys.geometry.core.designer.edge.Edge]`

List of all edges of the face.

property `Face.loops`: `beartype.typing.List[FaceLoop]`

List of all loops of the face.

Method detail

`Face.normal(u: float = 0.5, v: float = 0.5) → ansys.geometry.core.math.vector.UnitVector3D`

Get the normal direction to the face at certain proportional 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 proportional 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) →
beartype.typing.List[*ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve*]

Create isoparametric curves at the given proportional parameter.

Typically, only one curve is created, but if the face has a hole, it is possible that more than one curve is created.

Parameters

use_u_param

[bool] Whether the parameter is the u coordinate or v coordinate. If True, it is the u coordinate. If False, it is the v coordinate.

parameter

[float] Proportional [0-1] parameter to create the one or more curves at.

Returns

List[TrimmedCurve]

List of curves that were created.

SurfaceType

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

Bases: *enum.Enum*

Provides values for the surface types supported by the Geometry service.

Overview

Attributes

<code>SURFACTYPE_UNKNOWN</code>
<code>SURFACTYPE_PLANE</code>
<code>SURFACTYPE_CYLINDER</code>
<code>SURFACTYPE_CONE</code>
<code>SURFACTYPE_TORUS</code>
<code>SURFACTYPE_SPHERE</code>
<code>SURFACTYPE_NURBS</code>
<code>SURFACTYPE_PROCEDURAL</code>

Import detail

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

Attribute detail

```
SurfaceType.SURFACTYPE_UNKNOWN = 0
SurfaceType.SURFACTYPE_PLANE = 1
SurfaceType.SURFACTYPE_CYLINDER = 2
SurfaceType.SURFACTYPE_CONE = 3
SurfaceType.SURFACTYPE_TORUS = 4
SurfaceType.SURFACTYPE_SPHERE = 5
SurfaceType.SURFACTYPE_NURBS = 6
SurfaceType.SURFACTYPE_PROCEDURAL = 7
```

FaceLoopType

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

Bases: `enum.Enum`

Provides values for the face loop types supported by the Geometry service.

Overview

Attributes

<code>INNER_LOOP</code>
<code>OUTER_LOOP</code>

Import detail

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

Attribute detail

```
FaceLoopType.INNER_LOOP = 'INNER'
```

```
FaceLoopType.OUTER_LOOP = 'OUTER'
```

Description

Module for managing a face.

The `part.py` module

Summary

Classes

<code>Part</code>	Represents a part master.
<code>MasterComponent</code>	Represents a part occurrence.

Part

```
class ansys.geometry.core.designer.part.Part(id: str, name: str, components:
    beartype.typing.List[MasterComponent], bodies:
    beartype.typing.List[ansys.geometry.core.designer.body.MasterBody])
```

Represents a part master.

This class should not be accessed by users. The `Part` class holds fundamental data of an assembly.

Parameters

id
[str] Unique identifier for the part.

name
[str] Name of the part.

components

[List[MasterComponent]] List of MasterComponent children that the part contains.

bodies

[List[MasterBody]] List of MasterBody children that the part contains. These are master bodies.

Overview**Properties**

<i>id</i>	ID of the part.
<i>name</i>	Name of the part.
<i>components</i>	MasterComponent children that the part contains.
<i>bodies</i>	MasterBody children that the part contains.

Special methods

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

Import detail

```
from ansys.geometry.core.designer.part import Part
```

Property detail

property Part.**id**: str

ID of the part.

property Part.**name**: str

Name of the part.

property Part.**components**: beartype.typing.List[MasterComponent]

MasterComponent children that the part contains.

property Part.**bodies**: beartype.typing.List[ansys.geometry.core.designer.body.MasterBody]

MasterBody children that the part contains.

These are master bodies.

Method detail

Part.**__repr__**() → str

Represent the part as a string.

MasterComponent

```
class ansys.geometry.core.designer.part.MasterComponent(id: str, name: str, part: Part, transform:
ansys.geometry.core.math.matrix.Matrix44
= IDENTITY_MATRIX44)
```

Represents a part occurrence.

Parameters

id

[str] Unique identifier for the transformed part.

name

[str] Name of the transformed part.

part

[Part] Reference to the transformed part's master part.

transform

[Matrix44] 4x4 transformation matrix from the master part.

Notes

This class should not be accessed by users. It holds the fundamental data of an assembly. Master components wrap parts by adding a transform matrix.

Overview

Properties

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

Special methods

__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:`

beartype.typing.List`[ansys.geometry.core.designer.component.Component]`

List of all occurrences of the component.

property `MasterComponent.part: Part`

Master part of the transformed part.

property `MasterComponent.transform: ansys.geometry.core.math.matrix.Matrix44`

4x4 transformation matrix from the master part.

Method detail

`MasterComponent.__repr__()` → `str`

Represent the master component as a string.

Description

Module providing fundamental data of an assembly.

The selection.py module

Summary

Classes

<code>NamedSelection</code>	Represents a single named selection within the design assembly.
-----------------------------	---

NamedSelection

```
class ansys.geometry.core.designer.selection.NamedSelection(name: str, grpc_client: an-
sys.geometry.core.connection.client.GrpcClient,
bodies:
beartype.typing.Optional[beartype.typing.List[ansys.geo
= None, faces:
beartype.typing.Optional[beartype.typing.List[ansys.geo
= None, edges:
beartype.typing.Optional[beartype.typing.List[ansys.geo
= None, beams:
beartype.typing.Optional[beartype.typing.List[ansys.geo
= None, design_points:
beartype.typing.Optional[beartype.typing.List[ansys.geo
= None, preexisting_id:
beartype.typing.Optional[str] =
None)
```

Represents a single named selection within the design assembly.

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

A named selection organizes one or more design entities together for common actions against the entire group.

Parameters

name

[str] User-defined name for the named selection.

grpc_client

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

bodies

[List[Body], default: None] All bodies to include in the named selection.

faces

[List[Face], default: None] All faces to include in the named selection.

edges

[List[Edge], default: None] All edges to include in the named selection.

beams

[List[Beam], default: None] All beams to include in the named selection.

design_points

[List[DesignPoints], default: None] All design points to include in the named selection.

Overview

Properties

<i>id</i>	ID of the named selection.
<i>name</i>	Name of the named selection.

Import detail

```
from ansys.geometry.core.designer.selection import NamedSelection
```

Property detail

property `NamedSelection.id: str`

ID of the named selection.

property `NamedSelection.name: str`

Name of the named selection.

Description

Module for creating a named selection.

Description

PyAnsys Geometry designer subpackage.

The materials package

Summary

Submodules

<i>material</i>	Provides the data structure for material and for adding a material property.
<i>property</i>	Provides the MaterialProperty class.

The material.py module

Summary

Classes

<i>Material</i>	Provides the data structure for a material.
-----------------	---

Material

```
class ansys.geometry.core.materials.material.Material(name: str, density: pint.Quantity,
                                                    additional_properties:
                                                    beartype.typing.Optional[beartype.typing.Sequence[ansys.geom
                                                    = 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

<code>add_property</code>	Add a material property to the Material class.
---------------------------	--

Properties

<code>properties</code>	Dictionary of the material property type and material properties.
<code>name</code>	Material name.

Import detail

```
from ansys.geometry.core.materials.material import Material
```

Property detail

property Material.properties:

`beartype.typing.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 for adding a material property.

The `property.py` module

Summary

Classes

<code>MaterialProperty</code>	Provides the data structure for a material property.
-------------------------------	--

Enums

<code>MaterialPropertyType</code>	Provides an enum holding the possible values for <code>MaterialProperty</code> objects.
-----------------------------------	---

`MaterialProperty`

```
class ansys.geometry.core.materials.property.MaterialProperty(type:
    beartype.typing.Union[MaterialPropertyType,
    str], name: str, quantity:
    beartype.typing.Union[pint.Quantity,
    ansys.geometry.core.typing.Real])
```

Provides the data structure for a material property.

Parameters

- type**
[Union[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: Union[~pint.Quantity, Real]

Value and unit in case of a supported Quantity. If the type is not supported, it must be a Real value (float or integer).

Overview

Properties

<i>type</i>	Material property ID.
<i>name</i>	Material property name.
<i>quantity</i>	Material property quantity and unit.

Import detail

```
from ansys.geometry.core.materials.property import MaterialProperty
```

Property detail

property `MaterialProperty.type: beartype.typing.Union[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: beartype.typing.Union[pint.Quantity, ansys.geometry.core.typing.Real]`
Material property quantity and unit.

If the type is not supported, it will be a Real value (float or integer).

MaterialPropertyType

class `ansys.geometry.core.materials.property.MaterialPropertyType(*args, **kws)`
Bases: `enum.Enum`

Provides an enum holding the possible values for MaterialProperty objects.

Overview

Methods

<code>from_id</code>	Return the MaterialPropertyType value from the service representation.
----------------------	--

Attributes

<code>DENSITY</code>
<code>ELASTIC_MODULUS</code>
<code>POISSON_RATIO</code>
<code>SHEAR_MODULUS</code>
<code>SPECIFIC_HEAT</code>
<code>TENSILE_STRENGTH</code>
<code>THERMAL_CONDUCTIVITY</code>

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

`MaterialPropertyType.from_id()` → *MaterialPropertyType*

Return the MaterialPropertyType value from the service representation.

Parameters

id

[str] Geometry Service string representation of a property type.

Returns

MaterialPropertyType

Common name for property type.

Description

Provides the MaterialProperty class.

Description

PyAnsys Geometry materials subpackage.

The math package

Summary

Submodules

<i>bbox</i>	Provides for managing a bounding box.
<i>constants</i>	Provides mathematical constants.
<i>frame</i>	Provides for managing a frame.
<i>matrix</i>	Provides matrix primitive representations.
<i>plane</i>	Provides primitive representation of a 2D plane in 3D space.
<i>point</i>	Provides geometry primitive representation for 2D and 3D points.
<i>vector</i>	Provides for creating and managing 2D and 3D vectors.

The bbox.py module

Summary

Classes

<i>BoundingBox2D</i>	Maintains the X and Y dimensions.
----------------------	-----------------------------------

BoundingBox2D

```
class ansys.geometry.core.math.bbox.BoundingBox2D(x_min: ansys.geometry.core.typing.Real =
    sys.float_info.max, x_max:
    ansys.geometry.core.typing.Real =
    sys.float_info.min, y_min:
    ansys.geometry.core.typing.Real =
    sys.float_info.max, y_max:
    ansys.geometry.core.typing.Real =
    sys.float_info.min)
```

Maintains the X and Y dimensions.

Parameters

x_min

[Real] Minimum value for the x-dimensional bounds.

- x_max**
[Real] Maximum value for the x-dimensional bounds.
- y_min**
[Real] Minimum value for the y-dimensional bounds.
- y_max**
[Real] Maximum value for the y-dimensional bounds.

Overview

Methods

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

Properties

<code>x_min</code>	Minimum value of X-dimensional bounds.
<code>x_max</code>	Maximum value of the X-dimensional bounds.
<code>y_min</code>	Minimum value of Y-dimensional bounds.
<code>y_max</code>	Maximum value of Y-dimensional bounds.

Special methods

<code>__eq__</code>	Equals operator for the <code>BoundingBox2D</code> class.
<code>__ne__</code>	Not equals operator for the <code>BoundingBox2D</code> 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*: *beartype.typing.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 provided point lies within the X and Y ranges of the 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 current X and Y ranges of the bounds.

Parameters

x

[`Real`] Point X component to compare against the bounds.

y

[`Real`] Point Y component to compare against the bounds.

Returns

bool

True if the components are contained in the bounding box. Otherwise, `False`.

`BoundingBox2D.__eq__`(*other*: `BoundingBox2D`) → `bool`

Equals operator for the `BoundingBox2D` class.

`BoundingBox2D.__ne__`(*other*: `BoundingBox2D`) → `bool`

Not equals operator for the `BoundingBox2D` class.

Description

Provides for managing a bounding box.

The constants.py module

Summary

Constants

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

Description

Provides mathematical constants.

Module detail

`constants.DEFAULT_POINT3D`

Default value for a 3D point.

`constants.DEFAULT_POINT2D`

Default value for a 2D point.

`constants.IDENTITY_MATRIX33`

Identity for a `Matrix33` object.

`constants.IDENTITY_MATRIX44`

Identity for a `Matrix44` object.

`constants.UNITVECTOR3D_X`

Default 3D unit vector in the Cartesian traditional X direction.

`constants.UNITVECTOR3D_Y`

Default 3D unit vector in the Cartesian traditional Y direction.

`constants.UNITVECTOR3D_Z`

Default 3D unit vector in the Cartesian traditional Z direction.

`constants.UNITVECTOR2D_X`

Default 2D unit vector in the Cartesian traditional X direction.

constants.**UNITVECTOR2D_Y**

Default 2D unit vector in the Cartesian traditional Y direction.

constants.**ZERO_VECTOR3D**

Zero-valued Vector3D object.

constants.**ZERO_VECTOR2D**

Zero-valued Vector2D object.

constants.**ZERO_POINT3D**

Zero-valued Point3D object.

constants.**ZERO_POINT2D**

Zero-valued Point2D object.

The `frame.py` module

Summary

Classes

<i>Frame</i>	Primitive representation of a frame (an origin and three fundamental directions).
--------------	---

Frame

```
class ansys.geometry.core.math.frame.Frame(origin: beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.point.Point3D] =
ZERO_POINT3D, direction_x:
beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.vector.UnitVector3D,
ansys.geometry.core.math.vector.Vector3D] =
UNITVECTOR3D_X, direction_y:
beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.vector.UnitVector3D,
ansys.geometry.core.math.vector.Vector3D] =
UNITVECTOR3D_Y)
```

Primitive representation of a frame (an origin and three fundamental directions).

Parameters

origin

[Union[*ndarray*, RealSequence, Point3D], default: ZERO_POINT3D] Centered origin of the frame. The default is ZERO_POINT3D, which is the Cartesian origin.

direction_x

[Union[*ndarray*, RealSequence, UnitVector3D, Vector3D], default: UNITVECTOR3D_X] X-axis direction.

direction_y
 [Union[ndarray, RealSequence, UnitVector3D, Vector3D], default:
 UNITVECTOR3D_Y] Y-axis direction.

Overview

Methods

<i>transform_point2d_local_to_global</i>	Transform a 2D point to a global 3D point.
--	--

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Frame class.
<code>__ne__</code>	Not equals operator for the Frame class.

Import detail

```
from ansys.geometry.core.math.frame import Frame
```

Property detail

property Frame.*origin*: *ansys.geometry.core.math.point.Point3D*

Origin of the frame.

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

X-axis direction of the frame.

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

Y-axis direction of the frame.

property Frame.*direction_z*: *ansys.geometry.core.math.vector.UnitVector3D*

Z-axis direction of the frame.

property `Frame.global_to_local_rotation`: *ansys.geometry.core.math.matrix.Matrix33*

Global to local space transformation matrix.

Returns

Matrix33

3x3 matrix representing the transformation from global to local coordinate space, excluding origin translation.

property `Frame.local_to_global_rotation`: *ansys.geometry.core.math.matrix.Matrix33*

Local to global space transformation matrix.

Returns

Matrix33

3x3 matrix representing the transformation from local to global coordinate space.

property `Frame.transformation_matrix`: *ansys.geometry.core.math.matrix.Matrix44*

Full 4x4 transformation matrix.

Returns

Matrix44

4x4 matrix representing the transformation from global to local coordinate space.

Method detail

`Frame.transform_point2d_local_to_global` (*point*: *ansys.geometry.core.math.point.Point2D*) → *ansys.geometry.core.math.point.Point3D*

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

This method transforms a local, plane-contained `Point2D` object in the global coordinate system, thus representing it as a `Point3D` object.

Parameters

point

[`Point2D`] `Point2D` local object to express in global coordinates.

Returns

Point3D

Global coordinates for the 3D point.

`Frame.__eq__` (*other*: `Frame`) → `bool`

Equals operator for the `Frame` class.

`Frame.__ne__` (*other*: `Frame`) → `bool`

Not equals operator for the `Frame` class.

Description

Provides for managing a frame.

The `matrix.py` module

Summary

Classes

<code>Matrix</code>	Provides matrix primitive representation.
<code>Matrix33</code>	Provides 3x3 matrix primitive representation.
<code>Matrix44</code>	Provides 4x4 matrix primitive representation.

Constants

<code>DEFAULT_MATRIX33</code>	Default value of the 3x3 identity matrix for the <code>Matrix33</code> class.
<code>DEFAULT_MATRIX44</code>	Default value of the 4x4 identity matrix for the <code>Matrix44</code> 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 primitive representation.

Parameters

input

[Union[`ndarray`, `RealSequence`]] Matrix arguments as a `np.ndarray` class.

Overview

Methods

<code>determinant</code>	Get the determinant of the matrix.
<code>inverse</code>	Provide the inverse of the matrix.

Special methods

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

Import detail

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

Method detail

`Matrix.determinant()` → `ansys.geometry.core.typing.Real`

Get the determinant of the matrix.

`Matrix.inverse()` → `Matrix`

Provide the inverse of the matrix.

`Matrix.__mul__(other: beartype.typing.Union[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 primitive representation.

Parameters

input

[`Union[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 primitive representation.

Parameters

input

[Union[`ndarray`, `RealSequence`, `Matrix`], default: `DEFAULT_MATRIX44`] Matrix arguments as a `np.ndarray` class.

Import detail

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

Description

Provides matrix primitive representations.

Module detail

`matrix.DEFAULT_MATRIX33`

Default value of the 3x3 identity matrix for the `Matrix33` class.

`matrix.DEFAULT_MATRIX44`

Default value of the 4x4 identity matrix for the `Matrix44` class.

The `plane.py` module

Summary

Classes

<code>Plane</code>	Provides primitive representation of a 2D plane in 3D space.
--------------------	--

Plane

```
class ansys.geometry.core.math.plane.Plane(origin: beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.point.Point3D] =
ZERO_POINT3D, direction_x:
beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.vector.UnitVector3D,
ansys.geometry.core.math.vector.Vector3D] =
UNITVECTOR3D_X, direction_y:
beartype.typing.Union[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

[Union[[ndarray](#), RealSequence, Point3D], default: ZERO_POINT3D] Centered origin of the frame. The default is ZERO_POINT3D, which is the Cartesian origin.

direction_x

[Union[[ndarray](#), RealSequence, UnitVector3D, Vector3D], default: UNITVECTOR3D_X] X-axis direction.

direction_y

[Union[[ndarray](#), RealSequence, UnitVector3D, Vector3D], default: UNITVECTOR3D_Y] Y-axis direction.

Overview

Methods

<code>is_point_contained</code>	Check if a 3D point is contained in the plane.
---------------------------------	--

Properties

<code>normal</code>	Calculate the normal vector of the plane.
---------------------	---

Special methods

<code>__eq__</code>	Equals operator for the Plane class.
<code>__ne__</code>	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

<i>Point2D</i>	Provides geometry primitive representation for a 2D point.
<i>Point3D</i>	Provides geometry primitive representation for a 3D point.

Constants

<i>DEFAULT_POINT2D_VALUES</i>	Default values for a 2D point.
<i>DEFAULT_POINT3D_VALUES</i>	Default values for a 3D point.
<i>BASE_UNIT_LENGTH</i>	Default value for the length of the base unit.

Point2D

```
class ansys.geometry.core.math.point.Point2D(input: beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence] =
DEFAULT_POINT2D_VALUES, unit:
beartype.typing.Optional[pint.Unit] = None)
```

Bases: *numpy.ndarray*, *ansys.geometry.core.misc.units.PhysicalQuantity*

Provides geometry primitive representation for a 2D point.

Parameters

input

[Union[*ndarray*, RealSequence], default: *DEFAULT_POINT2D_VALUES*] Direction arguments, either as a *numpy.ndarray* class or as a RealSequence.

unit

[Unit, default: *DEFAULT_UNITS.LENGTH*] Units for defining 2D point values.

Overview

Methods

<i>unit</i>	Get the unit of the object.
<i>base_unit</i>	Get the base unit of the object.

Properties

<code>x</code>	X plane component value.
<code>y</code>	Y plane component value.

Special methods

<code>__eq__</code>	Equals operator for the <code>Point2D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Point2D</code> class.
<code>__add__</code>	Add operation for the <code>Point2D</code> class.
<code>__sub__</code>	Subtraction operation for the <code>Point2D</code> class.

Import detail

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

Property detail

property `Point2D.x`: `pint.Quantity`

X plane component value.

property `Point2D.y`: `pint.Quantity`

Y plane component value.

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: beartype.typing.Union[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: beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence] =
    DEFAULT_POINT3D_VALUES, unit:
    beartype.typing.Optional[pint.Unit] = None)
```

Bases: `numpy.ndarray`, `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides geometry primitive representation for a 3D point.

Parameters

input

[Union[`ndarray`, `RealSequence`], default: `DEFAULT_POINT3D_VALUES`] Direction arguments, either as a `numpy.ndarray` class or as a `RealSequence`.

unit

[`Unit`, default: `DEFAULT_UNITS.LENGTH`] Units for defining 3D point values.

Overview

Methods

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

Properties

<code>x</code>	X plane component value.
<code>y</code>	Y plane component value.
<code>z</code>	Z plane component value.

Special methods

<code>__eq__</code>	Equals operator for the <code>Point3D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Point3D</code> class.
<code>__add__</code>	Add operation for the <code>Point3D</code> class.
<code>__sub__</code>	Subtraction operation for the <code>Point3D</code> class.

Import detail

```
from ansys.geometry.core.math.point import Point3D
```

Property detail

property `Point3D.x`: `pint.Quantity`

X plane component value.

property `Point3D.y`: `pint.Quantity`

Y plane component value.

property `Point3D.z`: `pint.Quantity`

Z plane component value.

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: beartype.typing.Union[Point3D, ansys.geometry.core.math.vector.Vector3D]) → Point3D`

Add operation for the `Point3D` class.

`Point3D.__sub__(other: beartype.typing.Union[Point3D, ansys.geometry.core.math.vector.Vector3D]) → Point3D`

Subtraction operation for the `Point3D` class.

`Point3D.unit()` → `pint.Unit`

Get the unit of the object.

`Point3D.base_unit()` → `pint.Unit`

Get the base unit of the object.

`Point3D.transform(matrix: ansys.geometry.core.math.matrix.Matrix44) → Point3D`

Transform the 3D point with a transformation matrix.

Parameters

matrix

[`Matrix44`] 4x4 transformation matrix to apply to the point.

Returns

Point3D

New 3D point that is the transformed copy of the original 3D point after applying the transformation matrix.

Notes

Transform the `Point3D` object by applying the specified 4x4 transformation matrix and return a new `Point3D` object representing the transformed point.

Description

Provides geometry primitive representation for 2D and 3D points.

Module detail

`point.DEFAULT_POINT2D_VALUES`

Default values for a 2D point.

`point.DEFAULT_POINT3D_VALUES`

Default values for a 3D point.

`point.BASE_UNIT_LENGTH`

Default value for the length of the base unit.

The `vector.py` module

Summary

Classes

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

Vector3D

```
class ansys.geometry.core.math.vector.Vector3D(shape, dtype=float, buffer=None, offset=0,
                                             strides=None, order=None)
```

Bases: `numpy.ndarray`

Provides for managing and creating a 3D vector.

Parameters

input

[Union[`ndarray`, `RealSequence`]] 3D `numpy.ndarray` class with `shape(X)`.

Overview

Constructors

<i>from_points</i>	Create a 3D vector from two distinct 3D points.
--------------------	---

Methods

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

Properties

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

Special methods

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

Import detail

```
from ansys.geometry.core.math.vector import Vector3D
```

Property detail

property `Vector3D.x`: `ansys.geometry.core.typing.Real`

X coordinate of the `Vector3D` class.

property `Vector3D.y`: `ansys.geometry.core.typing.Real`

Y coordinate of the `Vector3D` class.

property `Vector3D.z`: `ansys.geometry.core.typing.Real`

Z coordinate of the `Vector3D` class.

property `Vector3D.norm`: `float`

Norm of the vector.

property `Vector3D.magnitude`: `float`

Norm of the vector.

property `Vector3D.is_zero`: `bool`

Check if all components of the 3D vector are zero.

Method detail

`Vector3D.is_perpendicular_to(other_vector: Vector3D) → bool`

Check if this vector and another vector are perpendicular.

`Vector3D.is_parallel_to(other_vector: Vector3D) → bool`

Check if this vector and another vector are parallel.

`Vector3D.is_opposite(other_vector: Vector3D) → bool`

Check if this vector and another vector are opposite.

`Vector3D.normalize() → Vector3D`

Return a normalized version of the 3D vector.

`Vector3D.transform(matrix: ansys.geometry.core.math.matrix.Matrix44) → Vector3D`

Transform the 3D vector3D with a transformation matrix.

Parameters

matrix

[`Matrix44`] 4x4 transformation matrix to apply to the vector.

Returns

Vector3D

A new 3D vector that is the transformed copy of the original 3D vector after applying the transformation matrix.

Notes

Transform the `Vector3D` object by applying the specified 4x4 transformation matrix and return a new `Vector3D` object representing the transformed vector.

`Vector3D.get_angle_between(v: Vector3D) → pint.Quantity`

Get the angle between this 3D vector and another 3D vector.

Parameters

v

[`Vector3D`] Other 3D vector for computing the angle.

Returns

Quantity

Angle between these two 3D vectors.

`Vector3D.cross(v: Vector3D) → Vector3D`

Get 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: beartype.typing.Union[Vector3D, ansys.geometry.core.typing.Real]) → beartype.typing.Union[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: beartype.typing.Union[Vector3D, ansys.geometry.core.math.point.Point3D]) → beartype.typing.Union[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: beartype.typing.Union[numpy.ndarray, ansys.geometry.core.typing.RealSequence, ansys.geometry.core.math.point.Point3D], point_b: beartype.typing.Union[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

[`Point3D`] `Point3D` class representing the first point.

point_b

[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**

[Union[`ndarray`, `RealSequence`]] 2D `numpy.ndarray` class with shape(X,).

Overview**Constructors**

<i>from_points</i>	Create a 2D vector from two distinct 2D points.
--------------------	---

Methods

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

Properties

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

Special methods

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

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: beartype.typing.Union[Vector2D, ansys.geometry.core.typing.Real]) → beartype.typing.Union[Vector2D, ansys.geometry.core.typing.Real]`

Overload * operator with dot product.

Notes

This method also admits scalar multiplication.

`Vector2D.__add__(other: beartype.typing.Union[Vector2D, ansys.geometry.core.math.point.Point2D]) → beartype.typing.Union[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) → Vector2D`

Overload % operator with cross product.

classmethod `Vector2D.from_points(point_a: beartype.typing.Union[numpy.ndarray, ansys.geometry.core.typing.RealSequence, ansys.geometry.core.math.point.Point2D], point_b: beartype.typing.Union[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
[Point2D] Point2D class representing the first point.

point_b

[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*, *Vector3D*]

- 1D *numpy.ndarray* class with shape(X,)
- *Vector3D*

Overview**Constructors**

<i>from_points</i> 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: beartype.typing.Union[numpy.ndarray,
                                                                    ansys.geometry.core.typing.RealSequence,
                                                                    ansys.geometry.core.math.point.Point3D], point_b:
                                                                    beartype.typing.Union[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
[Point3D] *Point3D* class representing the first point.

point_b
[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`, `Vector2D`]

- 1D `numpy.ndarray` class with shape(X,)
- `Vector2D`

Overview

Constructors

<code>from_points</code>	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: beartype.typing.Union[numpy.ndarray,
                                                                    ansys.geometry.core.typing.RealSequence,
                                                                    ansys.geometry.core.math.point.Point2D], point_b:
                                     beartype.typing.Union[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
[Point2D] *Point2D* class representing the first point.

point_b

[Point2D] *Point2D* class representing the second point.

Returns

UnitVector2D

2D unit vector from point_a to point_b.

Description

Provides for creating and managing 2D and 3D vectors.

Description

PyAnsys Geometry math subpackage.

The misc package

Summary

Submodules

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

The accuracy.py module

Summary

Classes

<i>Accuracy</i>	Provides decimal precision evaluations for actions such as equivalency.
-----------------	---

Constants

<i>LENGTH_ACCURACY</i>	Constant for decimal accuracy in length comparisons.
<i>ANGLE_ACCURACY</i>	Constant for decimal accuracy in angle comparisons.
<i>DOUBLE_ACCURACY</i>	Constant for double accuracy.

Accuracy

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

Provides decimal precision evaluations for actions such as equivalency.

Overview

Static methods

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

Parameters

a
[Real] First value.

b
[Real] Second value.

relative_tolerance
[Real] Relative tolerance accepted.

absolute_tolerance
[Real] Absolute tolerance accepted.

Returns

bool

True if the values are inside the accepted tolerances, False otherwise.

Description

Provides for evaluating decimal precision.

Module detail

`accuracy.LENGTH_ACCURACY: ansys.geometry.core.typing.Real = 1e-08`

Constant for decimal accuracy in length comparisons.

`accuracy.ANGLE_ACCURACY: ansys.geometry.core.typing.Real = 1e-06`

Constant for decimal accuracy in angle comparisons.

`accuracy.DOUBLE_ACCURACY: ansys.geometry.core.typing.Real = 1e-13`

Constant for double accuracy.

The auxiliary.py module

Summary

Functions

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

Description

Auxiliary functions for the PyAnsys Geometry library.

Module detail

`auxiliary.get_design_from_component`(*component*: `ansys.geometry.core.designer.component.Component`)
 → `ansys.geometry.core.designer.design.Design`

Get the Design of the given Component object.

Parameters

component

[Component] The component object for which to find the Design.

Returns

Design

The Design of the provided component object.

`auxiliary.get_design_from_body` (*body*: `ansys.geometry.core.designer.body.Body`) →
`ansys.geometry.core.designer.design.Design`

Get the Design of the given Body object.

Parameters**body**

[Body] The body object for which to find the Design.

Returns**Design**

The Design of the provided body object.

`auxiliary.get_design_from_face` (*face*: `ansys.geometry.core.designer.face.Face`) →
`ansys.geometry.core.designer.design.Design`

Get the Design of the given Face object.

Parameters**face**

[Face] The face object for which to find the Design.

Returns**Design**

The Design of the provided face object.

`auxiliary.get_design_from_edge` (*edge*: `ansys.geometry.core.designer.edge.Edge`) →
`ansys.geometry.core.designer.design.Design`

Get the Design of the given Edge object.

Parameters**edge**

[Edge] The edge object for which to find the Design.

Returns**Design**

The Design of the provided edge object.

`auxiliary.get_bodies_from_ids` (*design*: `ansys.geometry.core.designer.design.Design`, *body_ids*:
`beartype.typing.List[str]`) →
`beartype.typing.List[ansys.geometry.core.designer.body.Body]`

Find the Body objects inside a Design from its ids.

Parameters**design**

[Design] Parent design for the faces.

body_ids

[List[str]] List of body ids.

Returns**List[Body]**

List of Body objects.

Notes

This method takes a design and body ids, and gets their corresponding Body object.

```
auxiliary.get_faces_from_ids(design: ansys.geometry.core.designer.design.Design, face_ids:  
                             beartype.typing.List[str]) →  
                             beartype.typing.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:  
                             beartype.typing.List[str]) →  
                             beartype.typing.List[ansys.geometry.core.designer.edge.Edge]
```

Find the Edge objects inside a Design from its ids.

Parameters

design

[Design] Parent design for the edges.

edge_ids

[List[str]] List of edge ids.

Returns

List[Edge]

List of Edge objects.

Notes

This method takes a design and edge ids, and gets their corresponding Edge objects.

The checks.py module

Summary

Functions

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

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*: *beartype.typing.Optional[beartype.typing.Union[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*: *beartype.typing.Optional[beartype.typing.Union[str, None]] = None*) → *None*

Check if a `numpy.ndarray` has float or integer values.

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: beartype.typing.Optional[beartype.typing.Union[str, None]] = None) → None`

Check if a `numpy.ndarray` has all None values.

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: beartype.typing.Optional[beartype.typing.Union[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: beartype.typing.Optional[beartype.typing.Union[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 for `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: beartype.typing.Union[type, beartype.typing.Tuple[type, beartype.typing.Any]]) → None`

Check if an input object is of the same type as expected types.

Parameters**input**

[object] Input object.

expected_type

[Union[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: beartype.typing.Iterable, expected_type: beartype.typing.Union[type, beartype.typing.Tuple[type, beartype.typing.Any]]) → None`

Check if all elements in an iterable are of the same type as expected types.

Parameters**input**

[Iterable] Input iterable.

expected_type

[Union[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 method's 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.

The measurements.py module

Summary

Classes

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

Constants

<i>DEFAULT_UNITS</i>	PyAnsys Geometry default units object.
----------------------	--

SingletonMeta

class ansys.geometry.core.misc.measurements.SingletonMeta

Bases: type

Provides a thread-safe implementation of a singleton design pattern.

Overview

Special methods

<code>__call__</code>	Return a single instance of the class.
-----------------------	--

Import detail

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

Method detail

SingletonMeta.__call__(*args, **kwargs)

Return a single instance of the class.

Possible changes to the value of the `__init__` argument do not affect the returned instance.

DefaultUnitsClass

class ansys.geometry.core.misc.measurements.DefaultUnitsClass

Provides default units for the PyAnsys Geometry singleton design pattern.

Overview

Properties

<code>LENGTH</code>	Default length unit for PyAnsys Geometry.
<code>ANGLE</code>	Default angle unit for PyAnsys Geometry.
<code>SERVER_LENGTH</code>	Default length unit for supporting Geometry services for gRPC messages.
<code>SERVER_AREA</code>	Default area unit for supporting Geometry services for gRPC messages.
<code>SERVER_VOLUME</code>	Default volume unit for supporting Geometry services for gRPC messages.
<code>SERVER_ANGLE</code>	Default angle unit for supporting Geometry services for gRPC messages.

Import detail

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

Property detail

property `DefaultUnitsClass.LENGTH: pint.Unit`

Default length unit for PyAnsys Geometry.

property `DefaultUnitsClass.ANGLE: pint.Unit`

Default angle unit for PyAnsys Geometry.

property `DefaultUnitsClass.SERVER_LENGTH: pint.Unit`

Default length unit for supporting Geometry services 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 supporting Geometry services 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 supporting Geometry services 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 supporting Geometry services for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

Measurement

class `ansys.geometry.core.misc.measurements.Measurement` (*value:*
beartype.typing.Union[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

[Union[Real, Quantity]] Value of the measurement.

unit

[Unit] Units for the measurement.

dimensions

[Unit] Units for extracting the dimensions of the measurement. If `~pint.Unit.meter` is given, the dimension extracted is `[length]`.

Overview

Properties

<code>value</code> Value of the measurement.
--

Special methods

<code>__eq__</code> Equals operator for the Measurement class.
--

Import detail

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

Property detail

property `Measurement.value: pint.Quantity`

Value of the measurement.

Method detail

`Measurement.__eq__(other: Measurement) → bool`

Equals operator for the Measurement class.

Distance

```
class ansys.geometry.core.misc.measurements.Distance(value:
    beartype.typing.Union[ansys.geometry.core.typing.Real,
    pint.Quantity], unit:
    beartype.typing.Optional[pint.Unit] = None)
```

Bases: Measurement

Provides the Measurement subclass for holding a distance.

Parameters**value**

[Union[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:
    beartype.typing.Union[ansys.geometry.core.typing.Real,
    pint.Quantity], unit:
    beartype.typing.Optional[pint.Unit] = None)
```

Bases: Measurement

Provides the Measurement subclass for holding an angle.

Parameters**value**

[Union[Real, Quantity]] Value of the angle.

unit

[Unit, default: DEFAULT_UNITS.ANGLE] Units for the distance.

Import detail

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

Description

Provides various measurement-related classes.

Module detail

measurements.DEFAULT_UNITS

PyAnsys Geometry default units object.

The options.py module

Summary

Classes

<i>ImportOptions</i> Import options when opening a file.
--

ImportOptions

class ansys.geometry.core.misc.options.ImportOptions

Import options when opening a file.

Parameters

cleanup_bodies

[bool = False] Simplify geometry and clean up topology.

import_coordinate_systems

[bool = False] Import coordinate systems.

import_curves

[bool = False] Import curves.

import_hidden_components_and_geometry

[bool = False] Import hidden components and geometry.

import_names

[bool = False] Import names of bodies and curves.

import_planes

[bool = False] Import planes.

import_points

[bool = False] Import points.

Overview

Methods

<i>to_dict</i> Provide the dictionary representation of the ImportOptions class.
--

Attributes

<code>cleanup_bodies</code>
<code>import_coordinate_systems</code>
<code>import_curves</code>
<code>import_hidden_components_and_geometry</code>
<code>import_names</code>
<code>import_planes</code>
<code>import_points</code>

Import detail

```
from ansys.geometry.core.misc.options import ImportOptions
```

Attribute detail

`ImportOptions.cleanup_bodies: bool = False`

`ImportOptions.import_coordinate_systems: bool = False`

`ImportOptions.import_curves: bool = False`

`ImportOptions.import_hidden_components_and_geometry: bool = False`

`ImportOptions.import_names: bool = False`

`ImportOptions.import_planes: bool = False`

`ImportOptions.import_points: bool = False`

Method detail

`ImportOptions.to_dict()`

Provide the dictionary representation of the `ImportOptions` class.

Description

Provides various option classes.

The `units.py` module

Summary

Classes

<i>PhysicalQuantity</i>	Provides the base class for handling units throughout PyAnsys Geometry.
-------------------------	---

Constants

<i>UNITS</i>	Units manager.
--------------	----------------

PhysicalQuantity

```
class ansys.geometry.core.misc.units.PhysicalQuantity(unit: pint.Unit, expected_dimensions:
                                                    beartype.typing.Optional[pint.Unit] = None)
```

Provides the base class for handling units throughout PyAnsys Geometry.

Parameters

unit

[Unit] Units for the class.

expected_dimensions

[Unit, default: None] Units for the dimensionality of the physical quantity.

Overview

Properties

<i>unit</i>	Unit of the object.
<i>base_unit</i>	Base unit of the object.

Import detail

```
from ansys.geometry.core.misc.units import PhysicalQuantity
```

Property detail

property `PhysicalQuantity.unit`: `pint.Unit`

Unit of the object.

property `PhysicalQuantity.base_unit`: `pint.Unit`

Base unit of the object.

Description

Provides for handling units homogeneously throughout PyAnsys Geometry.

Module detail

`units.UNITS`

Units manager.

Description

Provides the PyAnsys Geometry miscellaneous subpackage.

The plotting package

Summary

Subpackages

<code>widgets</code>	Submodule providing widgets for the PyAnsys Geometry plotter.
----------------------	---

Submodules

<code>plotter</code>	Provides plotting for various PyAnsys Geometry objects.
<code>plotter_helper</code>	Provides a wrapper to aid in plotting.
<code>plotting_types</code>	Data types for plotting.
<code>trame_gui</code>	Module for using <code>trame</code> for visualization.

The widgets package

Summary

Submodules

<i>button</i>	Provides for implementing buttons in PyAnsys Geometry.
<i>displace_arrows</i>	Provides the displacement arrows widget for the PyVista plotter.
<i>measure</i>	Provides the ruler widget for the PyAnsys Geometry plotter.
<i>ruler</i>	Provides the ruler widget for the PyAnsys Geometry plotter.
<i>show_design_point</i>	Provides the ruler widget for the PyAnsys Geometry plotter.
<i>view_button</i>	Provides the view button widget for changing the camera view.
<i>widget</i>	Provides the abstract implementation of plotter widgets.

The button.py module

Summary

Classes

<i>Button</i>	Provides the abstract class for implementing buttons in PyAnsys Geometry.
---------------	---

Button

class `ansys.geometry.core.plotting.widgets.button.Button`(*plotter*: `pyvista.Plotter`, *button_config*: `tuple`)

Bases: `ansys.geometry.core.plotting.widgets.widget.PlotterWidget`

Provides the abstract class for implementing buttons in PyAnsys Geometry.

Parameters

plotter

[Plotter] Plotter to draw the buttons on.

button_config

[tuple] Tuple containing the position and the path to the icon of the button.

Notes

This class wraps the PyVista `add_checkbox_button_widget()` method.

Overview

Abstract methods

callback Get the functionality of the button, which is implemented by subclasses.

Methods

update Assign the image that represents the button.

Import detail

```
from ansys.geometry.core.plotting.widgets.button import Button
```

Method detail

abstract `Button.callback(state: bool) → None`

Get the functionality of the button, which is implemented by subclasses.

Parameters

state

[bool] Whether the button is active.

`Button.update()` → None

Assign the image that represents the button.

Description

Provides for implementing buttons in PyAnsys Geometry.

The `displace_arrows.py` module

Summary

Classes

DisplacementArrow Defines the arrow to draw and what it is to do.

Enums

CameraPanDirection Provides an enum with the available movement directions of the camera.

DisplacementArrow

```
class ansys.geometry.core.plotting.widgets.displace_arrows.DisplacementArrow(plotter:
                                                                    pyvista.Plotter,
                                                                    direction:
                                                                    CameraPanDi-
                                                                    rection)
```

Bases: `ansys.geometry.core.plotting.widgets.button.Button`

Defines the arrow to draw and what it is to do.

Parameters

plotter

[Plotter] Plotter to draw the buttons on.

direction

[CameraPanDirection] Direction that the camera is to move.

Overview

Methods

callback Move the camera in the direction defined by the button.

Import detail

```
from ansys.geometry.core.plotting.widgets.displace_arrows import DisplacementArrow
```

Method detail

`DisplacementArrow.callback(state: bool) → None`

Move the camera in the direction defined by the button.

Parameters

state

[bool] State of the button, which is inherited from PyVista. The value is True if the button is active. However, this parameter is unused by this callback method.

CameraPanDirection

```
class ansys.geometry.core.plotting.widgets.displace_arrows.CameraPanDirection(*args,  
                                                                              **kws)
```

Bases: `enum.Enum`

Provides an enum with the available movement directions of the camera.

Overview

Attributes

<i>XUP</i>
<i>XDOWN</i>
<i>YUP</i>
<i>YDOWN</i>
<i>ZUP</i>
<i>ZDOWN</i>

Import detail

```
from ansys.geometry.core.plotting.widgets.displace_arrows import CameraPanDirection
```

Attribute detail

```
CameraPanDirection.XUP = (0, 'upxarrow.png', (5, 170))
```

```
CameraPanDirection.XDOWN = (1, 'downarrow.png', (5, 130))
```

```
CameraPanDirection.YUP = (2, 'upyarrow.png', (35, 170))
```

```
CameraPanDirection.YDOWN = (3, 'downarrow.png', (35, 130))
```

```
CameraPanDirection.ZUP = (4, 'upzarrow.png', (65, 170))
```

```
CameraPanDirection.ZDOWN = (5, 'downarrow.png', (65, 130))
```

Description

Provides the displacement arrows widget for the PyVista plotter.

The `measure.py` module

Summary

Classes

<code>MeasureWidget</code>	Provides the measure widget for the PyAnsys Geometry Plotter class.
----------------------------	---

MeasureWidget

class `ansys.geometry.core.plotting.widgets.measure.MeasureWidget`(*plotter_helper*: PlotterHelper)

Bases: `ansys.geometry.core.plotting.widgets.widget.PlotterWidget`

Provides the measure widget for the PyAnsys Geometry Plotter class.

Parameters

plotter_helper

[PlotterHelper] Provides the plotter to add the measure widget to.

Overview

Methods

<code>callback</code>	Remove or add the measurement widget actor upon click.
<code>update</code>	Define the measurement widget button params.

Import detail

```
from ansys.geometry.core.plotting.widgets.measure import MeasureWidget
```

Method detail

`MeasureWidget.callback`(*state*: *bool*) → None

Remove or add the measurement widget actor upon click.

Parameters

state

[*bool*] State of the button, which is inherited from PyVista. The value is True if the button is active.

`MeasureWidget.update`() → None

Define the measurement widget button params.

Description

Provides the ruler widget for the PyAnsys Geometry plotter.

The ruler.py module

Summary

Classes

<i>Ruler</i>	Provides the ruler widget for the PyAnsys Geometry Plotter class.
--------------	---

Ruler

class `ansys.geometry.core.plotting.widgets.ruler.Ruler`(*plotter*: `pyvista.Plotter`)

Bases: `ansys.geometry.core.plotting.widgets.widget.PlotterWidget`

Provides the ruler widget for the PyAnsys Geometry Plotter class.

Parameters

plotter

[`Plotter`] Provides the plotter to add the ruler widget to.

Overview

Methods

<i>callback</i>	Remove or add the ruler widget actor upon click.
<i>update</i>	Define the configuration and representation of the ruler widget button.

Import detail

```
from ansys.geometry.core.plotting.widgets.ruler import Ruler
```

Method detail

`Ruler.callback`(*state*: `bool`) → `None`

Remove or add the ruler widget actor upon click.

Parameters

state

[`bool`] State of the button, which is inherited from PyVista. The value is `True` if the button is active.

Notes

This method provides a callback function for the ruler widget. It is called every time the ruler widget is clicked.

`Ruler.update()` → `None`

Define the configuration and representation of the ruler widget button.

Description

Provides the ruler widget for the PyAnsys Geometry plotter.

The `show_design_point.py` module

Summary

Classes

<code>ShowDesignPoints</code>	Provides the a button to hide/show DesignPoint objects in the plotter.
-------------------------------	--

ShowDesignPoints

`class ansys.geometry.core.plotting.widgets.show_design_point.ShowDesignPoints`(*plotter_helper*: PlotterHelper)

Bases: `ansys.geometry.core.plotting.widgets.widget.PlotterWidget`

Provides the a button to hide/show DesignPoint objects in the plotter.

Parameters

plotter_helper

[PlotterHelper] Provides the plotter to add the button to.

Overview

Methods

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

Import detail

```
from ansys.geometry.core.plotting.widgets.show_design_point import ShowDesignPoints
```

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 button.

Description

Provides the ruler widget for the PyAnsys Geometry plotter.

The view_button.py module

Summary

Classes

<i>ViewButton</i>	Provides for changing the view.
-------------------	---------------------------------

Enums

<i>ViewDirection</i>	Provides an enum with the available views.
----------------------	--

ViewButton

```
class ansys.geometry.core.plotting.widgets.view_button.ViewButton(plotter: pyvista.Plotter,  
                                                                direction: tuple)
```

Bases: *ansys.geometry.core.plotting.widgets.button.Button*

Provides for changing the view.

Parameters

plotter

[Plotter] Plotter to draw the buttons on.

direction

[ViewDirection] Direction of the view.

Overview**Methods**

<i>callback</i>	Change the view depending on button interaction.
-----------------	--

Import detail

```
from ansys.geometry.core.plotting.widgets.view_button import ViewButton
```

Method detail

ViewButton.**callback**(*state: bool*) → None

Change the view depending on button interaction.

Parameters**state**

[bool] State of the button, which is inherited from PyVista. The value is True if the button is active.

Raises**NotImplementedError**

Raised if the specified direction is not implemented.

ViewDirection

```
class ansys.geometry.core.plotting.widgets.view_button.ViewDirection(*args, **kwargs)
```

Bases: `enum.Enum`

Provides an enum with the available views.

Overview**Attributes**

<i>XYPLUS</i>
<i>XYMINUS</i>
<i>XZPLUS</i>
<i>XZMINUS</i>
<i>YZPLUS</i>
<i>YZMINUS</i>
<i>ISOMETRIC</i>

Import detail

```
from ansys.geometry.core.plotting.widgets.view_button import ViewDirection
```

Attribute detail

```
ViewDirection.XYPLUS = (0, '+xy.png', (5, 220))
ViewDirection.XYMINUS = (1, '-xy.png', (5, 251))
ViewDirection.XZPLUS = (2, '+xz.png', (5, 282))
ViewDirection.XZMINUS = (3, '-xz.png', (5, 313))
ViewDirection.YZPLUS = (4, '+yz.png', (5, 344))
ViewDirection.YZMINUS = (5, '-yz.png', (5, 375))
ViewDirection.ISOMETRIC = (6, 'isometric.png', (5, 406))
```

Description

Provides the view button widget for changing the camera view.

The widget.py module

Summary

Classes

<i>PlotterWidget</i> Provides an abstract class for plotter widgets.
--

PlotterWidget

```
class ansys.geometry.core.plotting.widgets.widget.PlotterWidget(plotter: pyvista.Plotter)
```

Bases: `abc.ABC`

Provides an abstract class for plotter widgets.

Parameters

plotter

[`Plotter`] Plotter instance to add the widget to.

Notes

These widgets are intended to be used with PyVista plotter objects. More specifically, the way in which this abstraction has been built ensures that these widgets are easily integrable with PyAnsys Geometry's own `Plotter` class.

Overview

Abstract methods

<code>callback</code>	General callback function for <code>PlotterWidget</code> objects.
<code>update</code>	General update function for <code>PlotterWidget</code> objects.

Properties

<code>plotter</code>	Plotter object the widget is assigned to.
----------------------	---

Import detail

```
from ansys.geometry.core.plotting.widgets.widget import PlotterWidget
```

Property detail

property `PlotterWidget.plotter`: `pyvista.Plotter`

Plotter object the widget is assigned to.

Method detail

abstract `PlotterWidget.callback(state)` → `None`

General callback function for `PlotterWidget` objects.

abstract `PlotterWidget.update()` → `None`

General update function for `PlotterWidget` objects.

Description

Provides the abstract implementation of plotter widgets.

Description

Submodule providing widgets for the PyAnsys Geometry plotter.

The `plotter.py` module

Summary

Classes

<code>Plotter</code>	Provides for plotting sketches and bodies.
----------------------	--

Constants

<code>DEFAULT_COLOR</code>	Default color we use for the plotter actors.
<code>PICKED_COLOR</code>	Color to use for the actors that are currently picked.
<code>EDGE_COLOR</code>	Default color to use for the edges.
<code>PICKED_EDGE_COLOR</code>	Color to use for the edges that are currently picked.

Plotter

```
class ansys.geometry.core.plotting.plotter.Plotter(scene: beartype.typing.Optional[pyvista.Plotter] = None, color_opts: beartype.typing.Optional[beartype.typing.Dict] = None, num_points: int = 100, enable_widgets: bool = True)
```

Provides for plotting sketches and bodies.

Parameters

scene

[`Plotter`, default: `None`] Scene instance for rendering the objects.

color_opts

[`dict`, default: `None`] Dictionary containing the background and top colors.

num_points

[`int`, default: 100] Number of points to use to render the shapes.

enable_widgets: bool, default: True

Whether to enable widget buttons in the plotter window. Widget buttons must be disabled when using `trame` for visualization.

Overview

Methods

<code>view_xy</code>	View the scene from the XY plane.
<code>view_xz</code>	View the scene from the XZ plane.
<code>view_yx</code>	View the scene from the YX plane.
<code>view_yz</code>	View the scene from the YZ plane.
<code>view_zx</code>	View the scene from the ZX plane.
<code>view_zy</code>	View the scene from the ZY plane.
<code>plot_frame</code>	Plot a frame in the scene.
<code>plot_plane</code>	Plot a plane in the scene.
<code>plot_sketch</code>	Plot a sketch in the scene.
<code>add_body_edges</code>	Add the outer edges of a body to the plot.
<code>add_body</code>	Add a body to the scene.
<code>add_component</code>	Add a component to the scene.
<code>add_sketch_polydata</code>	Add sketches to the scene from PyVista polydata.
<code>clip</code>	Clip the passed mesh with a plane.
<code>add_design_point</code>	Add a DesignPoint object to the plotter.
<code>add</code>	Add any type of object to the scene.
<code>add_list</code>	Add a list of any type of object to the scene.
<code>show</code>	Show the rendered scene on the screen.

Properties

<code>scene</code>	Rendered scene object.
<code>geom_object_actors_map</code>	Mapping between the ~pyvista.Actor and the PyAnsys Geometry objects.

Import detail

```
from ansys.geometry.core.plotting.plotter import Plotter
```

Property detail

property Plotter.`scene`: `pyvista.plotting.plotter.Plotter`

Rendered scene object.

Returns

`Plotter`

Rendered scene object.

property Plotter.`geom_object_actors_map`: `beartype.typing.Dict[pyvista.Actor, ansys.geometry.core.plotting.plotting_types.GeoObjectPlot]`

Mapping between the ~pyvista.Actor and the PyAnsys Geometry objects.

Method detail

`Plotter.view_xy()` → `None`

View the scene from the XY plane.

`Plotter.view_xz()` → `None`

View the scene from the XZ plane.

`Plotter.view_yx()` → `None`

View the scene from the YX plane.

`Plotter.view_yz()` → `None`

View the scene from the YZ plane.

`Plotter.view_zx()` → `None`

View the scene from the ZX plane.

`Plotter.view_zy()` → `None`

View the scene from the ZY plane.

`Plotter.plot_frame`(*frame*: `ansys.geometry.core.math.frame.Frame`, *plotting_options*: `beartype.typing.Optional[beartype.typing.Dict] = 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.

`Plotter.plot_plane`(*plane*: `ansys.geometry.core.math.plane.Plane`, *plane_options*: `beartype.typing.Optional[beartype.typing.Dict] = None`, *plotting_options*: `beartype.typing.Optional[beartype.typing.Dict] = 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.

`Plotter.plot_sketch`(*sketch*: `ansys.geometry.core.sketch.sketch.Sketch`, *show_plane*: `bool = False`, *show_frame*: `bool = False`, ***plotting_options*: `beartype.typing.Optional[beartype.typing.Dict]`) → `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.

`Plotter.add_body_edges`(*body_plot*: `ansys.geometry.core.plotting.plotting_types.GeoObjectPlot`,
***plotting_options*: `beartype.typing.Optional[dict]`) → `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**

[`GeomObjectPlot`] Body of which to add the edges.

****plotting_options**

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

`Plotter.add_body`(*body*: `ansys.geometry.core.designer.body.Body`, *merge*: `beartype.typing.Optional[bool] = False`, ***plotting_options*: `beartype.typing.Optional[beartype.typing.Dict]`) → `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.

`Plotter.add_component`(*component*: `ansys.geometry.core.designer.component.Component`, *merge_component*:
`bool = False`, *merge_bodies*: `bool = False`, ***plotting_options*) → `str`

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.

Returns**str**

Name of the added PyVista actor.

`Plotter.add_sketch_polydata`(*polydata_entries*: *beartype.typing.List[pyvista.PolyData]*, ***plotting_options*)
→ None

Add sketches to the scene from PyVista polydata.

Parameters**polydata**

[*pyvista.PolyData*] Polydata to add.

****plotting_options**

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

`Plotter.clip`(*mesh*: *beartype.typing.Union[pyvista.PolyData, pyvista.MultiBlock]*, *plane*:
ansys.geometry.core.math.plane.Plane = None) → *beartype.typing.Union[pyvista.PolyData,*
pyvista.MultiBlock]

Clip the passed mesh with a plane.

Parameters**mesh**

[*Union[pv.PolyData, pv.MultiBlock]*] Mesh you want to clip.

normal

[*str*, optional] Plane you want to use for clipping, by default "x". Available options: ["x", "-x", "y", "-y", "z", "-z"]

origin

[*tuple*, optional] Origin point of the plane, by default None

Returns**Union[pv.PolyData,pv.MultiBlock]**

The clipped mesh.

`Plotter.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.

`Plotter.add`(*object*: *beartype.typing.Any*, *merge_bodies*: *bool = False*, *merge_components*: *bool = False*, *filter*:
str = None, ***plotting_options*) → None

Add 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 that you want to plot.

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.

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.

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.

`Plotter.add_list(plotting_list: beartype.typing.List[beartype.typing.Any], merge_bodies: bool = False, merge_components: bool = False, 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.

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.

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.

`Plotter.show(show_axes_at_origin: bool = True, show_plane: bool = True, jupyter_backend: beartype.typing.Optional[str] = None, **kwargs: beartype.typing.Optional[beartype.typing.Dict]) → None`

Show the rendered scene on the screen.

Parameters**jupyter_backend**

[*str*, default: *None*] PyVista Jupyter backend.

****kwargs**

[*dict*, default: *None*] Plotting keyword arguments. For allowable keyword arguments, see the `Plotter.show` method.

Notes

For more information on supported Jupyter backends, see [Jupyter Notebook Plotting](#) in the PyVista documentation.

Description

Provides plotting for various PyAnsys Geometry objects.

Module detail

```
plotter.DEFAULT_COLOR = '#D6F7D1'
```

Default color we use for the plotter actors.

```
plotter.PICKED_COLOR = '#BB6EEE'
```

Color to use for the actors that are currently picked.

```
plotter.EDGE_COLOR = '#000000'
```

Default color to use for the edges.

```
plotter.PICKED_EDGE_COLOR = '#9C9C9C'
```

Color to use for the edges that are currently picked.

The `plotter_helper.py` module

Summary

Classes

<i>PlotterHelper</i> Provides for simplifying the selection of frame in <code>plot()</code> functions.
--

PlotterHelper

```
class ansys.geometry.core.plotting.plotter_helper.PlotterHelper(use_frame:  
                                                                beartype.typing.Optional[bool]  
                                                                = None, allow_picking:  
                                                                beartype.typing.Optional[bool]  
                                                                = False)
```

Provides for simplifying the selection of frame in `plot()` functions.

Parameters

`use_frame`

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

`allow_picking`: `bool`, default: `False`

Enables/disables the picking capabilities in the PyVista plotter.

Overview

Methods

<code>enable_widgets</code>	Enable the widgets for the plotter.
<code>select_object</code>	Select an object in the plotter.
<code>unselect_object</code>	Unselect an object in the plotter.
<code>picker_callback</code>	Define callback for the element picker.
<code>compute_edge_object_map</code>	Compute the mapping between plotter actors and <code>EdgePlot</code> objects.
<code>enable_picking</code>	Enable picking capabilities in the plotter.
<code>disable_picking</code>	Disable picking capabilities in the plotter.
<code>add</code>	Add a <code>pyansys-geometry</code> or <code>PyVista</code> object to the plotter.
<code>plot</code>	Plot and show any <code>PyAnsys Geometry</code> object.
<code>show_plotter</code>	Show the plotter or start the <code>trame</code> service.

Import detail

```
from ansys.geometry.core.plotting.plotter_helper import PlotterHelper
```

Method detail

`PlotterHelper.enable_widgets()`

Enable the widgets for the plotter.

`PlotterHelper.select_object(geom_object:`

beartype.typing.Union[ansys.geometry.core.plotting.plotting_types.GeoObjectPlot, ansys.geometry.core.plotting.plotting_types.EdgePlot], pt: numpy.ndarray) → None

Select an object in the plotter.

Highlights the object edges and adds a label with the object name and adds it to the `PyAnsys Geometry` object selection.

Parameters

geom_object

[Union[GeoObjectPlot, EdgePlot]] Geometry object to select.

pt

[ndarray] Set of points to determine the label position.

`PlotterHelper.unselect_object(geom_object:`

beartype.typing.Union[ansys.geometry.core.plotting.plotting_types.GeoObjectPlot, ansys.geometry.core.plotting.plotting_types.EdgePlot]) → None

Unselect an object in the plotter.

Removes edge highlighting and label from a plotter actor and removes it from the `PyAnsys Geometry` object selection.

Parameters

geom_object

[Union[GeoObjectPlot, EdgePlot]] Object to unselect.

`PlotterHelper.picker_callback(actor: pyvista.Actor) → None`

Define callback for the element picker.

Parameters

actor

[*Actor*] Actor that we are picking.

`PlotterHelper.compute_edge_object_map() → beartype.typing.Dict[pyvista.Actor,
ansys.geometry.core.plotting.plotting_types.EdgePlot]`

Compute the mapping between plotter actors and `EdgePlot` objects.

Returns

Dict[*Actor*, *EdgePlot*]

Mapping between plotter actors and `EdgePlot` objects.

`PlotterHelper.enable_picking()`

Enable picking capabilities in the plotter.

`PlotterHelper.disable_picking()`

Disable picking capabilities in the plotter.

`PlotterHelper.add(object: beartype.typing.Any, merge_bodies: bool = False, merge_component: bool = False,
filter: str = None, **plotting_options)`

Add a `pyansys-geometry` or `PyVista` object to the plotter.

Parameters

object

[*Any*] Object or list of objects you want to show.

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.

merge_component

[*bool*, default: `False`] Whether to merge this component into a single dataset. When `True`, all the individual bodies are effectively combined into a single dataset without any hierarchy.

****plotting_options**

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

`PlotterHelper.plot(object: beartype.typing.Any = None, screenshot: beartype.typing.Optional[str] = None,
merge_bodies: bool = False, merge_component: bool = False, view_2d:
beartype.typing.Dict = None, filter: str = None, **plotting_options) →
beartype.typing.List[beartype.typing.Any]`

Plot and show any `PyAnsys Geometry` object.

These types of objects are supported: `Body`, `Component`, `List[pv.PolyData]`, `pv.MultiBlock`, and `Sketch`.

Parameters

object

[*Any*, default: `None`] Any object or list of objects that you want to plot.

screenshot

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

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.

merge_component

[bool, default: `False`] Whether to merge this component into a single dataset. When True, all the individual bodies are effectively combined into a single dataset without any hierarchy.

view_2d

[Dict, default: `None`] Dictionary with the plane and the viewup vectors of the 2D plane.

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.

Returns**List[Any]**

List with the picked bodies in the picked order.

`PlotterHelper.show_plotter(screenshot: beartype.typing.Optional[str] = None) → None`

Show the plotter or start the `trame` service.

Parameters**plotter**

[Plotter] PyAnsys Geometry plotter with the meshes added.

screenshot

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

Description

Provides a wrapper to aid in plotting.

The `plotting_types.py` module**Summary****Classes**

<code>EdgePlot</code>	Mapper class to relate PyAnsys Geometry edges with its PyVista actor.
<code>GeomObjectPlot</code>	Mapper class to relate PyAnsys Geometry objects with its PyVista actor.

EdgePlot

```
class ansys.geometry.core.plotting.plotting_types.EdgePlot(actor: pyvista.Actor, edge_object: ansys.geometry.core.designer.edge.Edge, parent: GeomObjectPlot = None)
```

Mapper class to relate PyAnsys Geometry edges with its PyVista actor.

Parameters

actor

[Actor] PyVista actor that represents the edge.

edge_object

[Edge] PyAnsys Geometry edge that is represented by the PyVista actor.

parent

[GeomObjectPlot, optional] Parent PyAnsys Geometry object of this edge, by default None.

Overview

Properties

<i>actor</i>	Return PyVista actor of the object.
<i>edge_object</i>	Return the PyAnsys Geometry edge.
<i>parent</i>	Parent PyAnsys Geometry object of this edge.
<i>name</i>	Return the name of the edge.

Import detail

```
from ansys.geometry.core.plotting.plotting_types import EdgePlot
```

Property detail

property EdgePlot.actor: `pyvista.Actor`

Return PyVista actor of the object.

Returns

Actor

PyVista actor.

property EdgePlot.edge_object: `ansys.geometry.core.designer.edge.Edge`

Return the PyAnsys Geometry edge.

Returns

Edge

PyAnsys Geometry edge.

property `EdgePlot.parent: beartype.typing.Any`

Parent PyAnsys Geometry object of this edge.

Returns

Any

PyAnsys Geometry object.

property `EdgePlot.name: str`

Return the name of the edge.

Returns

str

Name of the edge.

GeomObjectPlot

class `ansys.geometry.core.plotting.plotting_types.GeomObjectPlot`(*actor: pyvista.Actor, object: beartype.typing.Any, edges: beartype.typing.List[EdgePlot] = None, add_body_edges: bool = True*)

Mapper class to relate PyAnsys Geometry objects with its PyVista actor.

Parameters

actor

[`pv.Actor`] PyVista actor that represents the pyansys-geometry object.

object

[`Any`] PyAnsys Geometry object that is represented.

edges

[`List[EdgePlot]`, optional] List of edges of the PyAnsys Geometry object, by default `None`.

add_body_edges: bool, optional

Flag to specify if you want to be able to add edges.

Overview

Properties

<code>actor</code>	Return the PyVista actor of the PyAnsys Geometry object.
<code>object</code>	Return the PyAnsys Geometry object.
<code>edges</code>	Return the list of edges associated to this PyAnsys Geometry object.
<code>name</code>	Return the name of this object.
<code>add_body_edges</code>	Return whether you want to be able to add edges.

Import detail

```
from ansys.geometry.core.plotting.plotting_types import GeomObjectPlot
```

Property detail

property `GeomObjectPlot.actor: pyvista.Actor`

Return the PyVista actor of the PyAnsys Geometry object.

Returns

Actor

Actor of the PyAnsys Geometry object.

property `GeomObjectPlot.object: beartype.typing.Any`

Return the PyAnsys Geometry object.

Returns

Any

PyAnsys Geometry object.

property `GeomObjectPlot.edges: beartype.typing.List[EdgePlot]`

Return the list of edges associated to this PyAnsys Geometry object.

Returns

List[EdgePlot]

List of the edges of this object.

property `GeomObjectPlot.name: str`

Return the name of this object.

Returns

str

Name of the object.

property `GeomObjectPlot.add_body_edges: bool`

Return whether you want to be able to add edges.

Returns

bool

Flag to add edges.

Description

Data types for plotting.

The `trame_gui.py` module

Summary

Classes

<code>TrameVisualizer</code>	Defines the trame layout view.
------------------------------	--------------------------------

TrameVisualizer

```
class ansys.geometry.core.plotting.trame_gui.TrameVisualizer(client_type: str = 'vue3')
```

Defines the trame layout view.

Parameters

`client_type`

[`str`, optional] Client type to use for the trame server. Options are 'vue2' and 'vue3'. Default is 'vue3'.

Overview

Methods

<code>set_scene</code>	Set the trame layout view and the mesh to show through the PyVista plotter.
<code>show</code>	Start the trame server and show the mesh.

Import detail

```
from ansys.geometry.core.plotting.trame_gui import TrameVisualizer
```

Method detail

```
TrameVisualizer.set_scene(plotter: beartype.typing.Union[pyvista.Plotter,
    ansys.geometry.core.plotting.plotter.Plotter,
    ansys.geometry.core.plotting.plotter_helper.PlotterHelper])
```

Set the trame layout view and the mesh to show through the PyVista plotter.

Parameters

`plotter`

[`Union[Plotter, Plotter, PlotterHelper]`] PyVista plotter or PyAnsys plotter to render.

```
TrameVisualizer.show()
```

Start the trame server and show the mesh.

Description

Module for using `trame` for visualization.

Description

Provides the PyAnsys Geometry plotting subpackage.

The shapes package

Summary

Subpackages

<i>curves</i>	Provides the PyAnsys Geometry curves subpackage.
<i>surfaces</i>	Provides the PyAnsys Geometry surface subpackage.

Submodules

<i>box_uv</i>	Provides the BoxUV class.
<i>parameterization</i>	Provides the parametrization-related classes.

The curves package

Summary

Submodules

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

The circle.py module

Summary

Classes

<i>Circle</i>	Provides 3D circle representation.
<i>CircleEvaluation</i>	Provides evaluation of a circle at a given parameter.

Circle

```
class ansys.geometry.core.shapes.curves.circle.Circle(origin:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence,
    ansys.geometry.core.math.point.Point3D],
    radius: beartype.typing.Union[pint.Quantity,
    an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real], reference:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence,
    ansys.geometry.core.math.vector.UnitVector3D,
    ansys.geometry.core.math.vector.Vector3D] =
    UNITVECTOR3D_X, axis:
    beartype.typing.Union[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

[Union[*ndarray*, RealSequence, Point3D]] Origin of the circle.

radius

[Union[Quantity, Distance, Real]] Radius of the circle.

reference

[Union[*ndarray*, RealSequence, UnitVector3D, Vector3D]] X-axis direction.

axis

[Union[*ndarray*, RealSequence, UnitVector3D, Vector3D]] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Test whether a parameter is within the parametric range of the curve.
<code>contains_point</code>	Test whether the point is contained by the curve.

Methods

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

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Circle class.
---------------------	---------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.circle import Circle
```

Property detail

property `Circle.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the circle.

property `Circle.radius`: `pint.Quantity`

Radius of the circle.

property `Circle.diameter`: `pint.Quantity`

Diameter of the circle.

property `Circle.perimeter`: `pint.Quantity`

Perimeter of the circle.

property `Circle.area`: `pint.Quantity`

Area of the circle.

property `Circle.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the circle.

property `Circle.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the circle.

property `Circle.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the circle.

Method detail

`Circle.__eq__(other: Circle) → bool`

Equals operator for the Circle class.

`Circle.evaluate(parameter: ansys.geometry.core.typing.Real) → CircleEvaluation`

Evaluate the circle at a given parameter.

Parameters

parameter

[Real] Parameter to evaluate the circle at.

Returns

CircleEvaluation

Resulting evaluation.

`Circle.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Circle`

Create a transformed copy of the circle based on a transformation matrix.

Parameters

matrix

[Matrix44] 4x4 transformation matrix to apply to the circle.

Returns

Circle

New circle that is the transformed copy of the original circle.

`Circle.mirrored_copy()` → *Circle*

Create a mirrored copy of the circle along the y-axis.

Returns

Circle

A new circle that is a mirrored copy of the original circle.

`Circle.project_point(point: ansys.geometry.core.math.point.Point3D)` → *CircleEvaluation*

Project a point onto the circle and evaluate the circle.

Parameters

point

[Point3D] Point to project onto the circle.

Returns

CircleEvaluation

Resulting evaluation.

`Circle.is_coincident_circle(other: Circle)` → `bool`

Determine if the circle is coincident with another.

Parameters

other

[Circle] Circle to determine coincidence with.

Returns

bool

True if this circle is coincident with the other, False otherwise.

`Circle.parameterization()` → *ansys.geometry.core.shapes.parameterization.Parameterization*

Get the parametrization of the circle.

The parameter of a circle specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

Returns

Parameterization

Information about how the circle is parameterized.

abstract `Circle.contains_param(param: ansys.geometry.core.typing.Real)` → `bool`

Test whether a parameter is within the parametric range of the curve.

abstract `Circle.contains_point(point: ansys.geometry.core.math.point.Point3D)` → `bool`

Test whether the 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

Methods

<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation.
<i>normal</i>	Normal to the circle.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the circle.

Properties

<i>circle</i>	Circle being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.

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.

Method detail

`CircleEvaluation.position()` → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns**Point3D**

Point that lies on the circle at this evaluation.

`CircleEvaluation.tangent()` → *ansys.geometry.core.math.vector.UnitVector3D*

Tangent of the evaluation.

Returns**UnitVector3D**

Tangent unit vector to the circle at this evaluation.

`CircleEvaluation.normal()` → *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the circle.

Returns**UnitVector3D**

Normal unit vector to the circle at this evaluation.

`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.

`CircleEvaluation.second_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

Second derivative of the evaluation.

Returns**Vector3D**

Second derivative of the evaluation.

`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

<code>Curve</code>	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

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

Methods

<code>trim</code>	Trim this curve by bounding it with an interval.
-------------------	--

Import detail

```
from ansys.geometry.core.shapes.curves.curve import Curve
```


Method detail

abstract `Curve.parameterization()` → *ansys.geometry.core.shapes.parameterization.Parameterization*

Parameterize the curve.

abstract `Curve.contains_param(param: ansys.geometry.core.typing.Real)` → `bool`

Test whether a parameter is within the parametric range of the curve.

abstract `Curve.contains_point(point: ansys.geometry.core.math.point.Point3D)` → `bool`

Test whether the point is contained by the curve.

The point can either lie within the curve or on its boundary.

abstract `Curve.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44)` → *Curve*

Create a transformed copy of the curve.

abstract `Curve.__eq__(other: Curve)` → `bool`

Determine if two curves are equal.

abstract `Curve.evaluate(parameter: ansys.geometry.core.typing.Real)` →
ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation

Evaluate the curve at the given parameter.

abstract `Curve.project_point(point: ansys.geometry.core.math.point.Point3D)` →
ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation

Project a point to the curve.

This method returns the evaluation at the closest point.

`Curve.trim(interval: ansys.geometry.core.shapes.parameterization.Interval)` →
ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve

Trim this curve by bounding it with an interval.

Returns

TrimmedCurve

The resulting bounded curve.

Description

Provides the `Curve` class.

The `curve_evaluation.py` module

Summary

Classes

<i>CurveEvaluation</i>	Provides for evaluating a curve.
------------------------	----------------------------------

CurveEvaluation

class ansys.geometry.core.shapes.curves.curve_evaluation.**CurveEvaluation**(parameter: ansys.geometry.core.typing.Real = None)

Provides for evaluating a curve.

Overview

Abstract methods

<i>position</i>	Position of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the evaluation.

Methods

<i>is_set</i>	Determine if the parameter for the evaluation has been set.
---------------	---

Properties

<i>parameter</i>	Parameter that the evaluation is based upon.
------------------	--

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.

Method detail

`CurveEvaluation.is_set()` → `bool`

Determine if the parameter for the evaluation has been set.

abstract `CurveEvaluation.position()` → `ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

abstract `CurveEvaluation.first_derivative()` → `ansys.geometry.core.math.vector.Vector3D`

First derivative of the evaluation.

abstract `CurveEvaluation.second_derivative()` → `ansys.geometry.core.math.vector.Vector3D`

Second derivative of the evaluation.

abstract `CurveEvaluation.curvature()` → `ansys.geometry.core.typing.Real`

Curvature of the evaluation.

Description

Provides for creating and managing a curve.

The `ellipse.py` module

Summary

Classes

<code>Ellipse</code>	Provides 3D ellipse representation.
<code>EllipseEvaluation</code>	Evaluate an ellipse at a given parameter.

Ellipse

```

class ansys.geometry.core.shapes.curves.ellipse.Ellipse(origin:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence,
    ansys.geometry.core.math.point.Point3D],
    major_radius:
    beartype.typing.Union[pint.Quantity, an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real],
    minor_radius:
    beartype.typing.Union[pint.Quantity, an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real],
    reference:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.vector.UnitVector3D,
    ansys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_X, axis:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.vector.UnitVector3D,
    ansys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_Z)

```

Bases: `ansys.geometry.core.shapes.curves.curve.Curve`

Provides 3D ellipse representation.

Parameters

origin

[Union[`ndarray`, `RealSequence`, `Point3D`]] Origin of the ellipse.

major_radius

[Union[`Quantity`, `Distance`, `Real`]] Major radius of the ellipse.

minor_radius

[Union[`Quantity`, `Distance`, `Real`]] Minor radius of the ellipse.

reference

[Union[`ndarray`, `RealSequence`, `UnitVector3D`, `Vector3D`]] X-axis direction.

axis

[Union[`ndarray`, `RealSequence`, `UnitVector3D`, `Vector3D`]] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Test whether a parameter is within the parametric range of the curve.
<code>contains_point</code>	Test whether the point is contained by the curve.

Methods

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

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Ellipse class.
---------------------	--

Import detail

```
from ansys.geometry.core.shapes.curves.ellipse import Ellipse
```

Property detail

property `Ellipse.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the ellipse.

property `Ellipse.major_radius`: `pint.Quantity`

Major radius of the ellipse.

property `Ellipse.minor_radius`: `pint.Quantity`

Minor radius of the ellipse.

property `Ellipse.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the ellipse.

property `Ellipse.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the ellipse.

property `Ellipse.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the ellipse.

property `Ellipse.eccentricity`: `ansys.geometry.core.typing.Real`

Eccentricity of the ellipse.

property `Ellipse.linear_eccentricity`: `pint.Quantity`

Linear eccentricity of the ellipse.

Notes

The linear eccentricity is the distance from the center to the focus.

property `Ellipse.semi_latus_rectum`: `pint.Quantity`

Semi-latus rectum of the ellipse.

property `Ellipse.perimeter`: `pint.Quantity`

Perimeter of the ellipse.

property `Ellipse.area`: `pint.Quantity`

Area of the ellipse.

Method detail

`Ellipse.__eq__(other: Ellipse) → bool`

Equals operator for the `Ellipse` class.

`Ellipse.mirrored_copy() → Ellipse`

Create a mirrored copy of the ellipse along the y-axis.

Returns

Ellipse

New ellipse that is a mirrored copy of the original ellipse.

`Ellipse.evaluate(parameter: ansys.geometry.core.typing.Real) → EllipseEvaluation`

Evaluate the ellipse at the given parameter.

Parameters

parameter
[Real] Parameter to evaluate the ellipse at.

Returns

EllipseEvaluation
Resulting evaluation.

`Ellipse.project_point(point: ansys.geometry.core.math.point.Point3D) → EllipseEvaluation`

Project a point onto the ellipse and evaluate the ellipse.

Parameters

point
[Point3D] Point to project onto the ellipse.

Returns

EllipseEvaluation
Resulting evaluation.

`Ellipse.is_coincident_ellipse(other: Ellipse) → bool`

Determine if this ellipse is coincident with another.

Parameters

other
[Ellipse] Ellipse to determine coincidence with.

Returns

bool
True if this ellipse is coincident with the other, False otherwise.

`Ellipse.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Ellipse`

Create a transformed copy of the ellipse based on a transformation matrix.

Parameters

matrix
[Matrix44] 4x4 transformation matrix to apply to the ellipse.

Returns

Ellipse
New ellipse that is the transformed copy of the original ellipse.

`Ellipse.parameterization() → ansys.geometry.core.shapes.parameterization.Parameterization`

Get the parametrization of the ellipse.

The parameter of an ellipse specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

Returns

Parameterization
Information about how the ellipse is parameterized.

abstract `Ellipse.contains_param`(*param*: `ansys.geometry.core.typing.Real`) → `bool`

Test whether a parameter is within the parametric range of the curve.

abstract `Ellipse.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Test whether the 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

Methods

<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation.
<i>normal</i>	Normal of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the ellipse.

Properties

<i>ellipse</i>	Ellipse being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.

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.

Method detail

EllipseEvaluation.position() → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the ellipse at this evaluation.

EllipseEvaluation.tangent() → *ansys.geometry.core.math.vector.UnitVector3D*

Tangent of the evaluation.

Returns

UnitVector3D

Tangent unit vector to the ellipse at this evaluation.

EllipseEvaluation.normal() → *ansys.geometry.core.math.vector.UnitVector3D*

Normal of the evaluation.

Returns

UnitVector3D

Normal unit vector to the ellipse at this evaluation.

EllipseEvaluation.first_derivative() → *ansys.geometry.core.math.vector.Vector3D*

First derivative of the evaluation.

The first derivative is in the direction of the tangent and has a magnitude equal to the velocity (rate of change of position) at that point.

Returns

Vector3D

First derivative of the evaluation.

EllipseEvaluation.second_derivative() → *ansys.geometry.core.math.vector.Vector3D*

Second derivative of the evaluation.

Returns

Vector3D

Second derivative of the evaluation.

`EllipseEvaluation.curvature()` → `ansys.geometry.core.typing.Real`

Curvature of the ellipse.

Returns

Real

Curvature of the ellipse.

Description

Provides for creating and managing an ellipse.

The `line.py` module

Summary

Classes

<code>Line</code>	Provides 3D line representation.
<code>LineEvaluation</code>	Provides for evaluating a line.

Line

```
class ansys.geometry.core.shapes.curves.line.Line(origin: beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.point.Point3D],
direction: beartype.typing.Union[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

[`Union[ndarray, RealSequence, Point3D]]` Origin of the line.

direction

[`Union[ndarray, RealSequence, UnitVector3D, Vector3D]]` Direction of the line.

Overview

Abstract methods

<code>contains_param</code>	Test whether a parameter is within the parametric range of the curve.
<code>contains_point</code>	Test whether the point is contained by the curve.

Methods

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

Properties

<code>origin</code>	Origin of the line.
<code>direction</code>	Direction of the line.

Special methods

<code>__eq__</code>	Equals operator for the Line class.
---------------------	-------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.line import Line
```

Property detail

property `Line.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the line.

property `Line.direction`: `ansys.geometry.core.math.vector.UnitVector3D`

Direction of the line.

Method detail

`Line.__eq__(other: object) → bool`

Equals operator for the Line class.

`Line.evaluate(parameter: float) → LineEvaluation`

Evaluate the line at a given parameter.

Parameters

parameter

[Real] Parameter to evaluate the line at.

Returns

LineEvaluation

Resulting evaluation.

`Line.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Line`

Create a transformed copy of the line based on a transformation matrix.

Parameters

matrix

[Matrix44] 4X4 transformation matrix to apply to the line.

Returns

Line

New line that is the transformed copy of the original line.

`Line.project_point(point: ansys.geometry.core.math.point.Point3D) → LineEvaluation`

Project a point onto the line and evaluate the line.

Parameters

point

[Point3D] Point to project onto the line.

Returns

LineEvaluation

Resulting evaluation.

`Line.is_coincident_line(other: Line) → bool`

Determine if the line is coincident with another line.

Parameters

other

[Line] Line to determine coincidence with.

Returns

bool

True if the line is coincident with another line, False otherwise.

`Line.is_opposite_line(other: Line) → bool`

Determine if the line is opposite another line.

Parameters

other

[Line] Line to determine opposition with.

Returns

bool

True if the line is opposite to another line.

Line.parameterization() → *ansys.geometry.core.shapes.parameterization.Parameterization*

Get the parametrization of the line.

The parameter of a line specifies the distance from the *origin* in the direction of *direction*.

Returns

Parameterization

Information about how the line is parameterized.

abstract Line.contains_param(*param: ansys.geometry.core.typing.Real*) → **bool**

Test whether a parameter is within the parametric range of the curve.

abstract Line.contains_point(*point: ansys.geometry.core.math.point.Point3D*) → **bool**

Test whether the 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

Methods

<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation, which is always equal to the direction of the line.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the line, which is always 0.

Properties

<i>line</i>	Line being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.

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.

Method detail

`LineEvaluation.position()` → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the line at this evaluation.

`LineEvaluation.tangent()` → *ansys.geometry.core.math.vector.UnitVector3D*

Tangent of the evaluation, which is always equal to the direction of the line.

Returns

UnitVector3D

Tangent unit vector to the line at this evaluation.

`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.

`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])`.

`LineEvaluation.curvature()` → `float`

Curvature of the line, which is always 0.

Returns

Real

Curvature of the line, which is always 0.

Description

Provides for creating and managing a line.

The `trimmed_curve.py` module

Summary

Classes

<code>TrimmedCurve</code>	Represents a trimmed curve.
<code>ReversedTrimmedCurve</code>	Represents a reversed trimmed curve.

TrimmedCurve

```
class ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve(geometry: an-
                                                                    sys.geometry.core.shapes.curves.curve.Curve,
                                                                    start: an-
                                                                    sys.geometry.core.math.point.Point3D,
                                                                    end: an-
                                                                    sys.geometry.core.math.point.Point3D,
                                                                    interval: an-
                                                                    sys.geometry.core.shapes.parameterization.Interval,
                                                                    length: pint.Quantity,
                                                                    grpc_client: an-
                                                                    sys.geometry.core.connection.client.GrpcClient
                                                                    = None)
```

Represents a trimmed curve.

A trimmed curve is a curve that has a boundary. This boundary comes in the form of an interval.

Parameters

geometry

[Curve] Underlying mathematical representation of the curve.

start

[Point3D] Start point of the curve.

end

[Point3D] End point of the curve.

interval

[Interval] Parametric interval that bounds the curve.

length

[Quantity] Length of the curve.

grpc_client

[GrpcClient, default: `None`] gRPC client that is required for advanced functions such as `intersect_curve()`.

Overview

Methods

<code>evaluate_proportion</code>	Evaluate the curve at a proportional value.
<code>intersect_curve</code>	Intersect this trimmed curve with another to receive the points of intersection.

Properties

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

Special methods

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

Import detail

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

Property detail

property `TrimmedCurve.geometry`: `ansys.geometry.core.shapes.curves.curve.Curve`
Underlying mathematical curve.

property `TrimmedCurve.start`: `ansys.geometry.core.math.point.Point3D`
Start point of the curve.

property `TrimmedCurve.end`: `ansys.geometry.core.math.point.Point3D`
End point of the curve.

property `TrimmedCurve.length`: `pint.Quantity`
Calculated length of the edge.

property `TrimmedCurve.interval`: `ansys.geometry.core.shapes.parameterization.Interval`
Interval of the curve that provides its boundary.

Method detail

`TrimmedCurve.evaluate_proportion(param: ansys.geometry.core.typing.Real) → ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate the curve at a proportional value.

A parameter of 0 corresponds to the start of the curve, while a parameter of 1 corresponds to the end of the curve.

Parameters

param
[Real] Parameter in the proportional range [0,1].

Returns

CurveEvaluation
Resulting curve evaluation.

`TrimmedCurve.intersect_curve(other: TrimmedCurve) → beartype.typing.List[ansys.geometry.core.math.point.Point3D]`

Intersect this trimmed curve with another to receive the points of intersection.

If the two trimmed curves do not intersect, an empty list is returned.

Parameters

other
[TrimmedCurve] Trimmed curve to intersect with.

Returns

List[Point3D]
All points of intersection between the curves.

`TrimmedCurve.__repr__() → str`
Represent the trimmed curve as a string.

ReversedTrimmedCurve

```
class ansys.geometry.core.shapes.curves.trimmed_curve.ReversedTrimmedCurve(geometry: an-
    sys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation,
    start: an-
    sys.geometry.core.math.point.Point3D,
    end: an-
    sys.geometry.core.math.point.Point3D,
    interval: an-
    sys.geometry.core.shapes.parameterization.parameterization.Interval,
    length:
    pint.Quantity,
    grpc_client: an-
    sys.geometry.core.connection.client.ConnectionClient,
    = 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

<code>evaluate_proportion</code>	Evaluate the curve at a proportional value.
----------------------------------	---

Import detail

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

Method detail

`ReversedTrimmedCurve.evaluate_proportion`(*param*: `ansys.geometry.core.typing.Real`) → `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate the curve at a proportional value.

A parameter of 0 corresponds to the start of the curve, while a parameter of 1 corresponds to the end of the curve.

Parameters

param

[Real] Parameter in the proportional range [0,1].

Returns

CurveEvaluation

Resulting curve evaluation.

Description

Trimmed curve class.

Description

Provides the PyAnsys Geometry curves subpackage.

The surfaces package

Summary

Submodules

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

The cone.py module

Summary

Classes

<i>Cone</i>	Provides 3D cone representation.
<i>ConeEvaluation</i>	Evaluate the cone at given parameters.

Cone

```

class ansys.geometry.core.shapes-surfaces-cone.Cone(origin: beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.point.Point3D],
radius: beartype.typing.Union[pint.Quantity,
ansys.geometry.core.misc.measurements.Distance,
ansys.geometry.core.typing.Real], half_angle:
beartype.typing.Union[pint.Quantity,
ansys.geometry.core.misc.measurements.Angle,
ansys.geometry.core.typing.Real], reference:
beartype.typing.Union[numpy.ndarray,
ansys.geometry.core.typing.RealSequence,
ansys.geometry.core.math.vector.UnitVector3D,
ansys.geometry.core.math.vector.Vector3D] =
UNITVECTOR3D_X, axis:
beartype.typing.Union[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

[Union[[ndarray](#), RealSequence, Point3D]] Origin of the cone.

radius

[Union[Quantity, Distance, Real]] Radius of the cone.

half_angle

[Union[Quantity, Angle, Real]] Half angle of the apex, determining the upward angle.

reference

[Union[[ndarray](#), RealSequence, UnitVector3D, Vector3D]] X-axis direction.

axis

[Union[[ndarray](#), RealSequence, UnitVector3D, Vector3D]] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Test whether a parameter is within the parametric range of the surface.
<code>contains_point</code>	Test whether the point is contained by the surface.

Methods

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

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Cone class.
---------------------	-------------------------------------

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 based on 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`() →

`beartype.typing.Tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the cone surface as a tuple (U and V respectively).

The U parameter specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the distance along the axis, with a zero parameter at the XY plane of the cone.

Returns

`Tuple[Parameterization, Parameterization]`

Information about how a cone's u and v parameters are parameterized, respectively.

abstract `Cone.contains_param`(*param_uv*: `ansys.geometry.core.shapes.parameterization.ParamUV`) → `bool`

Test whether a parameter is within the parametric range of the surface.

abstract `Cone.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Test whether the 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

Methods

<i>position</i>	Position of the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	Second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	Second derivative with respect to the V parameter.
<i>min_curvature</i>	Minimum curvature of the cone.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature of the cone.
<i>max_curvature_direction</i>	Maximum curvature direction.

Properties

<i>cone</i>	Cone being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.

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.

Method detail

ConeEvaluation.**position**() → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the cone at this evaluation.

ConeEvaluation.**normal**() → *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the cone at this evaluation.

`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.

`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.

`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.

`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.

`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.

`ConeEvaluation.min_curvature()` → *ansys.geometry.core.typing.Real*

Minimum curvature of the cone.

Returns

Real

Minimum curvature of the cone.

`ConeEvaluation.min_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Minimum curvature direction.

Returns

UnitVector3D

Minimum curvature direction.

`ConeEvaluation.max_curvature()` → *ansys.geometry.core.typing.Real*

Maximum curvature of the cone.

Returns

Real

Maximum curvature of the cone.

`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**

<i>Cylinder</i>	Provides 3D cylinder representation.
<i>CylinderEvaluation</i>	Provides evaluation of a cylinder at given parameters.

Cylinder

```

class ansys.geometry.core.shapes-surfaces.cylinder.Cylinder(origin:
    beartype.typing.Union[numpy.ndarray,
    an-
    sys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.point.Point3D],
    radius:
    beartype.typing.Union[pint.Quantity,
    an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real],
    reference:
    beartype.typing.Union[numpy.ndarray,
    an-
    sys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.vector.UnitVector3D,
    an-
    sys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_X, axis:
    beartype.typing.Union[numpy.ndarray,
    an-
    sys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.vector.UnitVector3D,
    an-
    sys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_Z)

```

Bases: `ansys.geometry.core.shapes-surfaces.surface.Surface`

Provides 3D cylinder representation.

Parameters

origin

[Union[ndarray, RealSequence, Point3D]] Origin of the cylinder.

radius

[Union[Quantity, Distance, Real]] Radius of the cylinder.

reference

[Union[ndarray, RealSequence, UnitVector3D, Vector3D]] X-axis direction.

axis

[Union[ndarray, RealSequence, UnitVector3D, Vector3D]] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Test whether a parameter is within the parametric range of the surface.
<code>contains_point</code>	Test whether the point is contained by the surface.

Methods

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

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Cylinder class.
---------------------	---

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*: *beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]*)
→ *pint.Quantity*

Get the surface area of the cylinder.

Parameters

height

[Union[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*: *beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]*) → *pint.Quantity*

Get the volume of the cylinder.

Parameters

height

[Union[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 based on 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()` →

`beartype.typing.Tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the cylinder surface as a tuple (U and V respectively).

The U parameter specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the distance along the axis, with a zero parameter at the XY plane of the cylinder.

Returns

Tuple[Parameterization, Parameterization]

Information about how a cylinder's u and v parameters are parameterized, respectively.

abstract `Cylinder.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV)` → `bool`

Test whether a parameter is within the parametric range of the surface.

abstract `Cylinder.contains_point(point: ansys.geometry.core.math.point.Point3D)` → `bool`

Test whether the 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: an-
                                                                    sys.geometry.core.shapes.parameterization.I
```

Bases: `ansys.geometry.core.shapes_surfaces_surface_evaluation.SurfaceEvaluation`

Provides evaluation of a cylinder at given parameters.

Parameters

cylinder: `~ansys.geometry.core.shapes_surfaces_cylinder.Cylinder`
Cylinder to evaluate.

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

Overview

Methods

<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	Second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	Second derivative with respect to the V parameter.
<code>min_curvature</code>	Minimum curvature of the cylinder.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature of the cylinder.
<code>max_curvature_direction</code>	Maximum curvature direction.

Properties

<code>cylinder</code>	Cylinder being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.

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.

Method detail

`CylinderEvaluation.position()` → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the cylinder at this evaluation.

`CylinderEvaluation.normal()` → *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the cylinder at this evaluation.

`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.

`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.

`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.

`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.

`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.

`CylinderEvaluation.min_curvature()` → *ansys.geometry.core.typing.Real*

Minimum curvature of the cylinder.

Returns

Real

Minimum curvature of the cylinder.

`CylinderEvaluation.min_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Minimum curvature direction.

Returns

UnitVector3D

Mminimum curvature direction.

`CylinderEvaluation.max_curvature()` → *ansys.geometry.core.typing.Real*

Maximum curvature of the cylinder.

Returns

Real

Maximum curvature of the cylinder.

`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

<i>PlaneSurface</i>	Provides 3D plane surface representation.
<i>PlaneEvaluation</i>	Provides evaluation of a plane at given parameters.

PlaneSurface

```
class ansys.geometry.core.shapes-surfaces.plane.PlaneSurface(origin:
    beartype.typing.Union[numpy.ndarray,
    an-
    sys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.point.Point3D],
    reference:
    beartype.typing.Union[numpy.ndarray,
    an-
    sys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.vector.UnitVector3D,
    an-
    sys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_X, axis:
    beartype.typing.Union[numpy.ndarray,
    an-
    sys.geometry.core.typing.RealSequence,
    an-
    sys.geometry.core.math.vector.UnitVector3D,
    an-
    sys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_Z)
```

Bases: `ansys.geometry.core.shapes-surfaces.surface.Surface`

Provides 3D plane surface representation.

Parameters

origin

[Union[ndarray, RealSequence, Point3D],] Centered origin of the plane.

reference

[Union[ndarray, RealSequence, UnitVector3D, Vector3D]] X-axis direction.

axis

[Union[ndarray, RealSequence, UnitVector3D, Vector3D]] X-axis direction.

Overview

Abstract methods

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

Methods

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

Properties

<i>origin</i>	Origin of the cylinder.
<i>dir_x</i>	X-direction of the cylinder.
<i>dir_y</i>	Y-direction of the cylinder.
<i>dir_z</i>	Z-direction of the cylinder.

Special methods

<code>__eq__</code>	Check whether two planes are equal.
---------------------	-------------------------------------

Import detail

```
from ansys.geometry.core.shapes-surfaces.plane import PlaneSurface
```

Property detail

property `PlaneSurface.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the cylinder.

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

X-direction of the cylinder.

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

Y-direction of the cylinder.

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

Z-direction of the cylinder.

Method detail

`PlaneSurface.__eq__(other: PlaneSurface) → bool`

Check whether two planes are equal.

abstract `PlaneSurface.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`

Test whether a ParamUV is within the parametric range of the surface.

abstract `PlaneSurface.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check whether a 3D point is in the domain of the plane.

`PlaneSurface.parameterization() → beartype.typing.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 matrix.

`PlaneSurface.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → PlaneEvaluation`

Evaluate the plane at a given u and v parameter.

PlaneEvaluation

class `ansys.geometry.core.shapes_surfaces_plane.PlaneEvaluation(plane: PlaneSurface, parameter: ansys.geometry.core.shapes.parameterization.ParamUV)`

Bases: `ansys.geometry.core.shapes_surfaces_surface_evaluation.SurfaceEvaluation`

Provides evaluation of a plane at given parameters.

Parameters

plane: `~ansys.geometry.core.shapes_surfaces_plane.PlaneSurface`
Plane to evaluate.

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

Overview

Methods

<i>position</i>	Point on the surface, based on the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to u.
<i>v_derivative</i>	First derivative with respect to v.
<i>uu_derivative</i>	Second derivative with respect to u.
<i>uv_derivative</i>	Second derivative with respect to u and v.
<i>vv_derivative</i>	Second derivative with respect to v.
<i>min_curvature</i>	Minimum curvature.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature.
<i>max_curvature_direction</i>	Maximum curvature direction.

Properties

<i>plane</i>	Plane being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.

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.

Method detail

`PlaneEvaluation.position()` → *ansys.geometry.core.math.point.Point3D*

Point on the surface, based on the evaluation.

`PlaneEvaluation.normal()` → *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

`PlaneEvaluation.u_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to u.

`PlaneEvaluation.v_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to v.

`PlaneEvaluation.uu_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to u.

`PlaneEvaluation.uv_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to u and v.

`PlaneEvaluation.vv_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to v.

`PlaneEvaluation.min_curvature()` → *ansys.geometry.core.typing.Real*

Minimum curvature.

`PlaneEvaluation.min_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Minimum curvature direction.

`PlaneEvaluation.max_curvature()` → *ansys.geometry.core.typing.Real*

Maximum curvature.

`PlaneEvaluation.max_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Maximum curvature direction.

Description

Provides for creating and managing a cylinder.

The `sphere.py` module

Summary

Classes

<i>Sphere</i>	Provides 3D sphere representation.
<i>SphereEvaluation</i>	Evaluate a sphere at given parameters.

Sphere

```
class ansys.geometry.core.shapes.surfaces.sphere.Sphere(origin:
    beartype.typing.Union[numpy.ndarray,
        ansys.geometry.core.typing.RealSequence,
        ansys.geometry.core.math.point.Point3D],
    radius:
    beartype.typing.Union[pint.Quantity, an-
        sys.geometry.core.misc.measurements.Distance,
        ansys.geometry.core.typing.Real],
    reference:
    beartype.typing.Union[numpy.ndarray,
        ansys.geometry.core.typing.RealSequence,
        an-
        sys.geometry.core.math.vector.UnitVector3D,
        ansys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_X, axis:
    beartype.typing.Union[numpy.ndarray,
        ansys.geometry.core.typing.RealSequence,
        an-
        sys.geometry.core.math.vector.UnitVector3D,
        ansys.geometry.core.math.vector.Vector3D]
    = UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes.surfaces.surface.Surface*

Provides 3D sphere representation.

Parameters

origin

[Union[*ndarray*, RealSequence, Point3D]] Origin of the sphere.

radius

[Union[Quantity, Distance, Real]] Radius of the sphere.

reference

[Union[*ndarray*, RealSequence, UnitVector3D, Vector3D]] X-axis direction.

axis

[Union[*ndarray*, RealSequence, UnitVector3D, Vector3D]] Z-axis direction.

Overview

Abstract methods

<i>contains_param</i>	Test whether a parameter is within the parametric range of the surface.
<i>contains_point</i>	Test whether the point is contained by the surface.

Methods

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

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Sphere class.
---------------------	---------------------------------------

Import detail

```
from ansys.geometry.core.shapes-surfaces.sphere import Sphere
```

Property detail

property Sphere.`origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the sphere.

property Sphere.`radius`: `pint.Quantity`

Radius of the sphere.

property Sphere.`dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the sphere.

property Sphere.`dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the sphere.

property Sphere.`dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the sphere.

property Sphere.`surface_area`: `pint.Quantity`

Surface area of the sphere.

property `Sphere.volume`: `pint.Quantity`

Volume of the sphere.

Method detail

`Sphere.__eq__(other: Sphere) → bool`

Equals operator for the `Sphere` class.

`Sphere.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Sphere`

Create a transformed copy of the sphere based on 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() →`

`beartype.typing.Tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterization of the sphere surface as a tuple (U and V respectively).

The U parameter specifies the longitude angle, increasing clockwise (east) about `dir_z` (right-hand corkscrew law). It has a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the latitude, increasing north, with a zero parameter at the equator and a range of $[-\pi/2, \pi/2]$.

Returns

Tuple[Parameterization, Parameterization]

Information about how a sphere's u and v parameters are parameterized, respectively.

abstract Sphere.**contains_param**(*param_uv*: ansys.geometry.core.shapes.parameterization.ParamUV) → bool

Test whether a parameter is within the parametric range of the surface.

abstract Sphere.**contains_point**(*point*: ansys.geometry.core.math.point.Point3D) → bool

Test whether the point is contained by the surface.

The point can either lie within the surface or on its boundary.

SphereEvaluation

class ansys.geometry.core.shapes-surfaces-sphere.SphereEvaluation(*sphere*: Sphere, *parameter*: ansys.geometry.core.shapes.parameterization.ParamUV)

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

Evaluate a sphere at given parameters.

Parameters

sphere: ~ansys.geometry.core.shapes-surfaces-sphere.Sphere
Sphere to evaluate.

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

Overview

Methods

<i>position</i>	Position of the evaluation.
<i>normal</i>	The normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	Second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	Second derivative with respect to the V parameter.
<i>min_curvature</i>	Minimum curvature of the sphere.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature of the sphere.
<i>max_curvature_direction</i>	Maximum curvature direction.

Properties

<i>sphere</i>	Sphere being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.

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.

Method detail

SphereEvaluation.**position**() → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the sphere at this evaluation.

SphereEvaluation.**normal**() → *ansys.geometry.core.math.vector.UnitVector3D*

The normal to the surface.

Returns

UnitVector3D

Normal unit vector to the sphere at this evaluation.

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.

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.

`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.

`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.

`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.

`SphereEvaluation.min_curvature()` → *ansys.geometry.core.typing.Real*

Minimum curvature of the sphere.

Returns

Real

Minimum curvature of the sphere.

`SphereEvaluation.min_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Minimum curvature direction.

Returns

UnitVector3D

Minimum curvature direction.

`SphereEvaluation.max_curvature()` → *ansys.geometry.core.typing.Real*

Maximum curvature of the sphere.

Returns

Real

Maximum curvature of the sphere.

`SphereEvaluation.max_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Maximum curvature direction.

Returns

UnitVector3D

Maximum curvature direction.

Description

Provides for creating and managing a sphere.

The `surface.py` module

Summary

Classes

<i>Surface</i>	Provides the abstract base class for a 3D surface.
----------------	--

Surface

```
class ansys.geometry.core.shapes-surfaces.surface.Surface
```

Bases: `abc.ABC`

Provides the abstract base class for a 3D surface.

Overview

Abstract methods

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

Methods

<i>trim</i>	Trim this surface by bounding it with a BoxUV.
-------------	--

Import detail

```
from ansys.geometry.core.shapes-surfaces.surface import Surface
```

Method detail

abstract `Surface.parameterization()` →
`beartype.typing.Tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the surface as a tuple (U and V respectively).

abstract `Surface.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`

Test whether a parameter is within the parametric range of the surface.

abstract `Surface.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Test whether the point is contained by the surface.

The point can either lie within the surface or on its boundary.

abstract `Surface.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Surface`

Create a transformed copy of the surface.

abstract `Surface.__eq__(other: Surface) → bool`

Determine if two surfaces are equal.

abstract `Surface.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation`

Evaluate the surface at the given parameter.

abstract `Surface.project_point(point: ansys.geometry.core.math.point.Point3D) → ansys.geometry.core.shapes_surfaces.surface_evaluation.SurfaceEvaluation`

Project a point to the surface.

This method returns the evaluation at the closest point.

`Surface.trim(box_uv: ansys.geometry.core.shapes.box_uv.BoxUV) → ansys.geometry.core.shapes_surfaces.trimmed_surface.TrimmedSurface`

Trim this surface by bounding it with a BoxUV.

Returns

TrimmedSurface

The resulting bounded surface.

Description

Provides the `Surface` class.

The `surface_evaluation.py` module

Summary

Classes

<code>SurfaceEvaluation</code>	Provides for evaluating a surface.
--------------------------------	------------------------------------

SurfaceEvaluation

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

an-
sys.geometry.core.shapes.param

Provides for evaluating a surface.

Overview

Abstract methods

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

Properties

<i>parameter</i>	Parameter that the evaluation is based upon.
------------------	--

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.

Method detail

abstract `SurfaceEvaluation.position()` → *ansys.geometry.core.math.point.Point3D*

Point on the surface, based on the evaluation.

abstract `SurfaceEvaluation.normal()` → *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

abstract `SurfaceEvaluation.u_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the U parameter.

abstract `SurfaceEvaluation.v_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the V parameter.

abstract `SurfaceEvaluation.uu_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U parameter.

abstract `SurfaceEvaluation.uv_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

The second derivative with respect to the U and V parameters.

abstract `SurfaceEvaluation.vv_derivative()` → *ansys.geometry.core.math.vector.Vector3D*

The second derivative with respect to v.

abstract `SurfaceEvaluation.min_curvature()` → *ansys.geometry.core.typing.Real*

Minimum curvature.

abstract `SurfaceEvaluation.min_curvature_direction()` →

ansys.geometry.core.math.vector.UnitVector3D

Minimum curvature direction.

abstract `SurfaceEvaluation.max_curvature()` → *ansys.geometry.core.typing.Real*

Maximum curvature.

abstract `SurfaceEvaluation.max_curvature_direction()` →

ansys.geometry.core.math.vector.UnitVector3D

Maximum curvature direction.

Description

Provides for evaluating a surface.

The `torus.py` module

Summary

Classes

<code>Torus</code>	Provides 3D torus representation.
<code>TorusEvaluation</code>	Evaluate the torus`` at given parameters.

Torus

```
class ansys.geometry.core.shapes-surfaces.torus.Torus(origin:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence,
    ansys.geometry.core.math.point.Point3D],
    major_radius:
    beartype.typing.Union[pint.Quantity, an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real],
    minor_radius:
    beartype.typing.Union[pint.Quantity, an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real], reference:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence, an-
    sys.geometry.core.math.vector.UnitVector3D,
    ansys.geometry.core.math.vector.Vector3D] =
    UNITVECTOR3D_X, axis:
    beartype.typing.Union[numpy.ndarray,
    ansys.geometry.core.typing.RealSequence, an-
    sys.geometry.core.math.vector.UnitVector3D,
    ansys.geometry.core.math.vector.Vector3D] =
    UNITVECTOR3D_Z)
```

Bases: `ansys.geometry.core.shapes-surfaces.surface.Surface`

Provides 3D torus representation.

Parameters

origin

[Union[`ndarray`, `RealSequence`, `Point3D`],] Centered origin of the torus.

direction_x

[Union[`ndarray`, `RealSequence`, `UnitVector3D`, `Vector3D`]] X-axis direction.

direction_y

[Union[`ndarray`, `RealSequence`, `UnitVector3D`, `Vector3D`]] Y-axis direction.

major_radius

[Union[`Quantity`, `Distance`, `Real`]] Major radius of the torus.

minor_radius

[Union[`Quantity`, `Distance`, `Real`]] Minor radius of the torus.

Overview

Abstract methods

<code>contains_param</code>	Test whether a parameter is within the parametric range of the surface.
<code>contains_point</code>	Test whether the point is contained by the surface.

Methods

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

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Torus class.
---------------------	--------------------------------------

Import detail

```
from ansys.geometry.core.shapes-surfaces.torus import Torus
```

Property detail

property `Torus.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the torus.

property `Torus.major_radius`: `pint.Quantity`

Semi-major radius of the torus.

property `Torus.minor_radius`: `pint.Quantity`

Semi-minor radius of the torus.

property `Torus.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the torus.

property `Torus.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the torus.

property `Torus.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the torus.

property `Torus.volume`: `pint.Quantity`

Volume of the torus.

property `Torus.surface_area`: `pint.Quantity`

Surface_area of the torus.

Method detail

`Torus.__eq__(other: Torus) → bool`

Equals operator for the Torus class.

`Torus.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Torus`

Create a transformed copy of the torus based on 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() →`

`beartype.typing.Tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the torus surface as a tuple (U and V respectively).

The U parameter specifies the longitude angle, increasing clockwise (east) about the axis (right-hand corkscrew law). It has a zero parameter at `Geometry.Frame.DirX` and a period of 2π .

The V parameter specifies the latitude, increasing north, with a zero parameter at the equator. For the donut, where the major radius is greater than the minor radius, the range is $[-\pi, \pi]$ and the parameterization is periodic. For a degenerate torus, the range is restricted accordingly and the parameterization is non-periodic.

Returns

Tuple[Parameterization, Parameterization]

Information about how a torus's u and v parameters are parameterized, respectively.

`Torus.project_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `TorusEvaluation`

Project a point onto the torus and evaluate the torus.

Parameters

point

[`Point3D`] Point to project onto the torus.

Returns

TorusEvaluation

Resulting evaluation.

abstract `Torus.contains_param`(*param_uv*: `ansys.geometry.core.shapes.parameterization.ParamUV`) → `bool`

Test whether a parameter is within the parametric range of the surface.

abstract `Torus.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `bool`

Test whether the point is contained by the surface.

The point can either lie within the surface or on its boundary.

TorusEvaluation

class `ansys.geometry.core.shapes_surfaces_torus.TorusEvaluation`(*torus*: `Torus`, *parameter*: `ansys.geometry.core.shapes.parameterization.ParamUV`)

Bases: `ansys.geometry.core.shapes_surfaces_surface_evaluation.SurfaceEvaluation`

Evaluate the torus`` at given parameters.

Parameters

Torus: `~ansys.geometry.core.shapes_surfaces_torus.Torus`

Torus to evaluate.

parameter: `ParamUV`

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

Overview

Methods

<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	Second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	Second derivative with respect to the V parameter.
<code>curvature</code>	Curvature of the torus.
<code>min_curvature</code>	Minimum curvature of the torus.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature of the torus.
<code>max_curvature_direction</code>	Maximum curvature direction.

Properties

<code>torus</code>	Torus being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.

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.

Method detail

`TorusEvaluation.position()` → *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the torus at this evaluation.

`TorusEvaluation.normal()` → *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the torus at this evaluation.

`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.

`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.

`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.

`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.

`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.

`TorusEvaluation.curvature()` → *beartype.typing.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.

`TorusEvaluation.min_curvature()` → *ansys.geometry.core.typing.Real*

Minimum curvature of the torus.

Returns

Real

Minimum curvature of the torus.

`TorusEvaluation.min_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Minimum curvature direction.

Returns

UnitVector3D

Minimum curvature direction.

`TorusEvaluation.max_curvature()` → *ansys.geometry.core.typing.Real*

Maximum curvature of the torus.

Returns

Real

Maximum curvature of the torus.

`TorusEvaluation.max_curvature_direction()` → *ansys.geometry.core.math.vector.UnitVector3D*

Maximum curvature direction.

Returns

UnitVector3D

Maximum curvature direction.

Description

Provides for creating and managing a torus.

The `trimmed_surface.py` module

Summary

Classes

<i>TrimmedSurface</i>	Represents a trimmed surface.
<i>ReversedTrimmedSurface</i>	Represents a reversed trimmed surface.

TrimmedSurface

class `ansys.geometry.core.shapes-surfaces-trimmed_surface.TrimmedSurface` (*geometry*: `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

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

Properties

<i>geometry</i>	Underlying mathematical surface.
<i>box_uv</i>	Bounding BoxUV of the surface.

Import detail

```
from ansys.geometry.core.shapes_surfaces_trimmed_surface import TrimmedSurface
```

Property detail

property `TrimmedSurface.geometry`: *ansys.geometry.core.shapes_surfaces_surface.Surface*
Underlying mathematical surface.

property `TrimmedSurface.box_uv`: *ansys.geometry.core.shapes_box_uv.BoxUV*
Bounding BoxUV of the surface.

Method detail

`TrimmedSurface.get_proportional_parameters`(*param_uv*:
ansys.geometry.core.shapes.parameterization.ParamUV) →
ansys.geometry.core.shapes.parameterization.ParamUV

Convert non-proportional parameters into proportional parameters.

Parameters

param_uv
[ParamUV] Non-proportional UV parameters.

Returns

ParamUV
Proportional (from 0-1) UV parameters.

`TrimmedSurface.normal`(*u*: *ansys.geometry.core.typing.Real*, *v*: *ansys.geometry.core.typing.Real*) →
ansys.geometry.core.math.vector.UnitVector3D

Provide the normal to the surface.

Parameters

u
[Real] First coordinate of the 2D representation of a surface in UV space.

v
[Real] Second coordinate of the 2D representation of a surface in UV space.

Returns

UnitVector3D
Unit vector is normal to the surface at the given UV coordinates.

`TrimmedSurface.project_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`

Project a point onto the surface and evaluate it at that location.

Parameters

point
[Point3D] Point to project onto the surface.

Returns

SurfaceEvaluation
Resulting evaluation.

`TrimmedSurface.evaluate_proportion`(*u*: `ansys.geometry.core.typing.Real`, *v*: `ansys.geometry.core.typing.Real`) → `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`

Evaluate the surface at proportional u and v parameters.

Parameters

u
[Real] U parameter in the proportional range [0,1].

v
[Real] V parameter in the proportional range [0,1].

Returns

SurfaceEvaluation
Resulting surface evaluation.

ReversedTrimmedSurface

```
class ansys.geometry.core.shapes-surfaces-trimmed_surface.ReversedTrimmedSurface(geometry:
    an-
    sys.geometry.core.shapes-surfaces-trimmed_surface.Surface,
    box_uv:
    an-
    sys.geometry.core.shapes-surfaces-box_uv.BoxUV):
```

Bases: `TrimmedSurface`

Represents a reversed trimmed surface.

When a surface is reversed, its normal vector is negated to provide the proper outward facing vector.

Parameters

face
[Face] Face that the trimmed surface belongs to.

geometry
[Surface] Underlying mathematical representation of the surface.

Overview

Methods

<i>normal</i>	Provide the normal to the surface.
<i>project_point</i>	Project a point onto the surface and evaluate it at that location.

Import detail

```
from ansys.geometry.core.shapes-surfaces-trimmed-surface import ReversedTrimmedSurface
```

Method detail

`ReversedTrimmedSurface.normal`(*u*: *ansys.geometry.core.typing.Real*, *v*: *ansys.geometry.core.typing.Real*) → *ansys.geometry.core.math.vector.UnitVector3D*

Provide the normal to the surface.

Parameters

u

[Real] First coordinate of the 2D representation of a surface in UV space.

v

[Real] Second coordinate of the 2D representation of a surface in UV space.

Returns

UnitVector3D

Unit vector is normal to the surface at the given UV coordinates.

`ReversedTrimmedSurface.project_point`(*point*: *ansys.geometry.core.math.point.Point3D*) → *ansys.geometry.core.shapes-surfaces-surface-evaluation.SurfaceEvaluation*

Project a point onto the surface and evaluate it at that location.

Parameters

point

[Point3D] Point to project onto the surface.

Returns

SurfaceEvaluation

Resulting evaluation.

Description

Provides the `TrimmedSurface` class.

Description

Provides the PyAnsys Geometry surface subpackage.

The `box_uv.py` module

Summary

Classes

<i>BoxUV</i>	Provides the implementation for <code>BoxUV</code> class.
--------------	---

Enums

<i>LocationUV</i>	Provides the enumeration for indicating locations for <code>BoxUV</code> .
-------------------	--

`BoxUV`

```
class ansys.geometry.core.shapes.box_uv.BoxUV(range_u:
    ansys.geometry.core.shapes.parameterization.Interval =
    None, range_v:
    ansys.geometry.core.shapes.parameterization.Interval =
    None)
```

Provides the implementation for `BoxUV` class.

Overview

Methods

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

Properties

<code>interval_u</code>	u interval.
<code>interval_v</code>	v interval.

Special methods

<code>__eq__</code>	Check whether two BoxUV instances are equal.
<code>__ne__</code>	Check whether two BoxUV instances are not equal.

Import detail

```
from ansys.geometry.core.shapes.box_uv import BoxUV
```

Property detail

property `BoxUV.interval_u`: *ansys.geometry.core.shapes.parameterization.Interval*
u interval.

property `BoxUV.interval_v`: *ansys.geometry.core.shapes.parameterization.Interval*
v interval.

Method detail

`BoxUV.__eq__(other: object) → bool`
Check whether two BoxUV instances are equal.

`BoxUV.__ne__(other: object) → bool`
Check whether two BoxUV instances are not equal.

`BoxUV.is_empty()`
Check if this BoxUV is empty.

`BoxUV.proportion(prop_u: ansys.geometry.core.typing.Real, prop_v: ansys.geometry.core.typing.Real) → ansys.geometry.core.shapes.parameterization.ParamUV`
Evaluate the BoxUV at the given proportions.

`BoxUV.get_center()` → *ansys.geometry.core.shapes.parameterization.ParamUV*
Evaluate the this BoxUV in the center.

`BoxUV.is_negative(tolerance_u: ansys.geometry.core.typing.Real, tolerance_v: ansys.geometry.core.typing.Real) → bool`
Check whether the BoxUV is negative.

`BoxUV.contains(param: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`
Check whether the BoxUV contains a given u and v pair parameter.

`BoxUV.inflate(delta_u: ansys.geometry.core.typing.Real, delta_v: ansys.geometry.core.typing.Real) → BoxUV`

Enlarge the BoxUV u and v intervals by delta_u and delta_v respectively.

`BoxUV.get_corner(location: LocationUV) → ansys.geometry.core.shapes.parameterization.ParamUV`

Get the corner location of the BoxUV.

LocationUV

`class ansys.geometry.core.shapes.box_uv.LocationUV(*args, **kws)`

Bases: `enum.Enum`

Provides the enumeration for indicating locations for BoxUV.

Overview

Attributes

<code>TopLeft</code>
<code>TopCenter</code>
<code>TopRight</code>
<code>BottomLeft</code>
<code>BottomCenter</code>
<code>BottomRight</code>
<code>LeftCenter</code>
<code>RightCenter</code>
<code>Center</code>

Import detail

```
from ansys.geometry.core.shapes.box_uv import LocationUV
```

Attribute detail

`LocationUV.TopLeft = 1`

`LocationUV.TopCenter = 2`

`LocationUV.TopRight = 3`

`LocationUV.BottomLeft = 4`

`LocationUV.BottomCenter = 5`

`LocationUV.BottomRight = 6`

`LocationUV.LeftCenter = 7`

`LocationUV.RightCenter = 8`

`LocationUV.Center = 9`

Description

Provides the BoxUV class.

The parameterization.py module

Summary

Classes

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

Enums

<i>ParamForm</i>	ParamForm enum class that defines the form of a Parameterization.
<i>ParamType</i>	ParamType enum class that defines the type of a Parameterization.

ParamUV

```
class ansys.geometry.core.shapes.parameterization.ParamUV(u: ansys.geometry.core.typing.Real, v: ansys.geometry.core.typing.Real)
```

Parameter class containing 2 parameters: (u, v).

Parameters

- u**
[Real] u-parameter.
- v**
[Real] v-parameter.

Notes

Likened to a 2D point in UV space Used as an argument in parametric surface evaluations. This matches the service implementation for the Geometry service.

Overview

Properties

<code>u</code>	u-parameter.
<code>v</code>	v-parameter.

Special methods

<code>__add__</code>	Add the u and v components of the other ParamUV to this ParamUV.
<code>__sub__</code>	Subtract the u and v components of the other ParamUV from this ParamUV.
<code>__mul__</code>	Multiplies the u and v components of this ParamUV by the other ParamUV.
<code>__truediv__</code>	Divides the u and v components of this ParamUV by the other ParamUV.
<code>__iter__</code>	Iterate a ParamUV.
<code>__repr__</code>	Represent the ParamUV as a string.

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamUV
```

Property detail

property ParamUV.u: `ansys.geometry.core.typing.Real`
u-parameter.

property ParamUV.v: `ansys.geometry.core.typing.Real`
v-parameter.

Method detail

ParamUV.`__add__`(*other*: ParamUV) → ParamUV

Add the u and v components of the other ParamUV to this ParamUV.

Parameters

other

[ParamUV] The parameters to add these parameters.

Returns

ParamUV

The sum of the parameters.

ParamUV.`__sub__`(*other*: ParamUV) → ParamUV

Subtract the u and v components of the other 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 the other 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 the other 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

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

Properties

<i>start</i>	Start value of the interval.
<i>end</i>	End value of the interval.

Static methods

<i>unite</i>	Get the union of two intervals.
<i>intersect</i>	Get the intersection of two intervals.

Special methods

<i>__eq__</i>	Compare two intervals.
<i>__repr__</i>	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.

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`

Boolean value that indicates 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 interval.

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* given the accuracy range.

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 `length_accuracy`.

Parameters

t
[Real] Value of interest.

Returns

bool
True if the interval contains the value, False otherwise.

`Interval.inflate`(*delta*: *ansys.geometry.core.typing.Real*) → *Interval*

Enlarge the current interval by the given delta value.

`Interval.__repr__`() → `str`

Represent the interval as a string.

Parameterization

`class` `ansys.geometry.core.shapes.parameterization.Parameterization`(*form*: `ParamForm`, *type*: `ParamType`, *interval*: `Interval`)

Parameterization class describes the parameters of a specific geometry.

Parameters

form
[ParamForm] Form of the parameterization.

type
[ParamType] Type of the parameterization.

interval
[Interval] Interval of the parameterization.

Overview

Properties

<i>form</i>	The form of the parameterization.
<i>type</i>	The type of the parameterization.
<i>interval</i>	The interval of the parameterization.

Special methods

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

Import detail

```
from ansys.geometry.core.shapes.parameterization import Parameterization
```

Property detail

property `Parameterization.form: ParamForm`

The form of the parameterization.

property `Parameterization.type: ParamType`

The type of the parameterization.

property `Parameterization.interval: Interval`

The interval of the parameterization.

Method detail

`Parameterization.__repr__()` → str

Represent the Parameterization as a string.

ParamForm

class `ansys.geometry.core.shapes.parameterization.ParamForm(*args, **kwargs)`

Bases: `enum.Enum`

ParamForm enum class that defines the form of a Parameterization.

Overview

Attributes

<i>OPEN</i>
<i>CLOSED</i>
<i>PERIODIC</i>
<i>OTHER</i>

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamForm
```

Attribute detail

`ParamForm.OPEN = 1`

`ParamForm.CLOSED = 2`

`ParamForm.PERIODIC = 3`

`ParamForm.OTHER = 4`

ParamType

```
class ansys.geometry.core.shapes.parameterization.ParamType(*args, **kwargs)
```

Bases: `enum.Enum`

ParamType enum class that defines the type of a Parameterization.

Overview

Attributes

<i>LINEAR</i>
<i>CIRCULAR</i>
<i>OTHER</i>

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamType
```

Attribute detail

ParamType.LINEAR = 1

ParamType.CIRCULAR = 2

ParamType.OTHER = 3

Description

Provides the parametrization-related classes.

Description

Provides the PyAnsys Geometry geometry subpackage.

The sketch package

Summary

Submodules

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

The `arc.py` module

Summary

Classes

<code>Arc</code> Provides for modeling an arc.
--

Arc

```
class ansys.geometry.core.sketch.arc.Arc(center: ansys.geometry.core.math.point.Point2D, start:
ansys.geometry.core.math.point.Point2D, end:
ansys.geometry.core.math.point.Point2D, clockwise:
beartype.typing.Optional[bool] = False)
```

Bases: `ansys.geometry.core.sketch.edge.SketchEdge`

Provides for modeling an arc.

Parameters

center

[Point2D] Center point of the arc.

start

[Point2D] Starting point of the arc.

end

[Point2D] Ending point of the arc.

clockwise

[bool, default: `False`] Whether the arc spans the clockwise angle between the start and end points. When `False` (default), the arc spans the counter-clockwise angle. When `True`, the arc spans the clockwise angle.

Overview

Constructors

<code>from_three_points</code> Create an arc from three given points.

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the Arc class.
<code>__ne__</code>	Not equals operator for the Arc class.

Import detail

```
from ansys.geometry.core.sketch.arc import Arc
```

Property detail

property Arc.start: *ansys.geometry.core.math.point.Point2D*

Starting point of the arc line.

property Arc.end: *ansys.geometry.core.math.point.Point2D*

Ending point of the arc line.

property Arc.length: *pint.Quantity*

Length of the arc.

property Arc.radius: *pint.Quantity*

Radius of the arc.

property Arc.center: *ansys.geometry.core.math.point.Point2D*

Center point 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.

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:  
    beartype.typing.Union[pint.Quantity,  
    ansys.geometry.core.misc.measurements.Distance,  
    ansys.geometry.core.typing.Real], height:  
    beartype.typing.Union[pint.Quantity,  
    ansys.geometry.core.misc.measurements.Distance,  
    ansys.geometry.core.typing.Real], angle:  
    beartype.typing.Optional[beartype.typing.Union[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

[Union[Quantity, Distance, Real]] Width of the box.

height

[Union[Quantity, Distance, Real]] Height of the box.

angle

[Union[Quantity, Angle, Real], default: 0] Placement angle for orientation alignment.

Overview

Properties

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

Import detail

```
from ansys.geometry.core.sketch.box import Box
```

Property detail

property `Box.center`: `ansys.geometry.core.math.point.Point2D`

Center point of the box.

property `Box.width`: `pint.Quantity`

Width of the box.

property `Box.height`: `pint.Quantity`

Height of the box.

property `Box.perimeter`: `pint.Quantity`

Perimeter of the box.

property `Box.area`: `pint.Quantity`

Area of the box.

property `Box.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Description

Provides for creating and managing a box (quadrilateral).

The `circle.py` module

Summary

Classes

<code>SketchCircle</code> Provides for modeling a circle.

SketchCircle

```
class ansys.geometry.core.sketch.circle.SketchCircle(center:
    ansys.geometry.core.math.point.Point2D,
    radius: beartype.typing.Union[pint.Quantity,
    an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real], plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```

Bases: *ansys.geometry.core.sketch.face.SketchFace*, *ansys.geometry.core.shapes.curves.circle.Circle*

Provides for modeling a circle.

Parameters

center: Point2D

Center point of the circle.

radius

[Union[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

<i>plane_change</i>	Redefine the plane containing the SketchCircle objects.
---------------------	---

Properties

<i>center</i>	Center of the circle.
<i>perimeter</i>	Perimeter of the circle.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.circle import SketchCircle
```

Property detail

property `SketchCircle.center`: `ansys.geometry.core.math.point.Point2D`

Center of the circle.

property `SketchCircle.perimeter`: `pint.Quantity`

Perimeter of the circle.

Notes

This property resolves the dilemma between using the `SkethFace.perimeter` property and the `Circle.perimeter` property.

property `SketchCircle.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Method detail

`SketchCircle.plane_change`(*plane*: `ansys.geometry.core.math.plane.Plane`) → `None`

Redefine the plane containing the `SketchCircle` objects.

Parameters

plane

[`Plane`] Desired new plane that is to contain the sketched circle.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing a circle.

The `edge.py` module

Summary

Classes

<code>SketchEdge</code> Provides for modeling edges forming sketched shapes.
--

SketchEdge

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

Provides for modeling edges forming sketched shapes.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing SketchEdge objects.
---------------------------	---

Properties

<code>start</code>	Starting point of the edge.
<code>end</code>	Ending point of the edge.
<code>length</code>	Length of the edge.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.edge import SketchEdge
```

Property detail

property `SketchEdge.start`: `ansys.geometry.core.math.point.Point2D`

Abstractmethod

Starting point of the edge.

property `SketchEdge.end`: `ansys.geometry.core.math.point.Point2D`

Abstractmethod

Ending point of the edge.

property `SketchEdge.length`: `pint.Quantity`

Abstractmethod

Length of the edge.

property `SketchEdge.visualization_polydata`: `pyvista.PolyData`

Abstractmethod

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData
 VTK pyvista.Polydata configuration.

Method detail

`SketchEdge.plane_change(plane: ansys.geometry.core.math.plane.Plane) → None`

Redefine the plane containing `SketchEdge` objects.

Parameters

plane
 [Plane] Desired new plane that is to contain the sketched edge.

Notes

This implies that their 3D definition might suffer changes. By default, this method does nothing. It is required to be implemented in child `SketchEdge` classes.

Description

Provides for creating and managing an edge.

The ellipse.py module

Summary

Classes

<i>SketchEllipse</i>	Provides for modeling an ellipse.
----------------------	-----------------------------------

SketchEllipse

```
class ansys.geometry.core.sketch.ellipse.SketchEllipse(center:
    ansys.geometry.core.math.point.Point2D,
    major_radius:
    beartype.typing.Union[pint.Quantity, an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real],
    minor_radius:
    beartype.typing.Union[pint.Quantity, an-
    sys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real], angle:
    beartype.typing.Optional[beartype.typing.Union[pint.Quantity,
    an-
    sys.geometry.core.misc.measurements.Angle,
    ansys.geometry.core.typing.Real]] = 0,
    plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```


Bases: `ansys.geometry.core.sketch.face.SketchFace`, `ansys.geometry.core.shapes.curves.ellipse.Ellipse`

Provides for modeling an ellipse.

Parameters

center: Point2D

Center point of the ellipse.

major_radius

[Union[Quantity, Distance, Real]] Major radius of the ellipse.

minor_radius

[Union[Quantity, Distance, Real]] Minor radius of the ellipse.

angle

[Union[Quantity, Angle, Real], default: 0] Placement angle for orientation alignment.

plane

[Plane, optional] Plane containing the sketched ellipse, which is the global XY plane by default.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing <code>SketchEllipse</code> objects.
---------------------------	---

Properties

<code>center</code>	Center point of the ellipse.
<code>angle</code>	Orientation angle of the ellipse.
<code>perimeter</code>	Perimeter of the circle.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.ellipse import SketchEllipse
```

Property detail

property `SketchEllipse.center`: `ansys.geometry.core.math.point.Point2D`

Center point of the ellipse.

property `SketchEllipse.angle`: `pint.Quantity`

Orientation angle of the ellipse.

property `SketchEllipse.perimeter`: `pint.Quantity`

Perimeter of the circle.

Notes

This property resolves the dilemma between using the `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

<i>SketchFace</i>	Provides for modeling a face.
-------------------	-------------------------------

SketchFace

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

Provides for modeling a face.

Overview

Methods

<i>plane_change</i>	Redefine the plane containing SketchFace objects.
---------------------	---

Properties

<i>edges</i>	List of all component edges forming the face.
<i>perimeter</i>	Perimeter of the face.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.face import SketchFace
```

Property detail

property SketchFace.**edges**:

beartype.typing.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**

<i>Gear</i>	Provides the base class for sketching gears.
<i>DummyGear</i>	Provides the dummy class for sketching gears.
<i>SpurGear</i>	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: beartype.typing.Union[pint.Quantity,  
anys.geometry.core.misc.measurements.Distance,  
anys.geometry.core.typing.Real], inner_radius:  
beartype.typing.Union[pint.Quantity,  
anys.geometry.core.misc.measurements.Distance,  
anys.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

[`Union[Quantity, Distance, Real]`] Outer radius of the gear.

inner_radius

[`Union[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: beartype.typing.Union[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

[Union[Quantity, Angle, Real]] Pressure angle of the spur gear.

n_teeth

[int] Number of teeth of the spur gear.

Overview

Properties

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

Import detail

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

Property detail

property `SpurGear.origin`: `ansys.geometry.core.math.point.Point2D`

Origin of the spur gear.

property `SpurGear.module`: `ansys.geometry.core.typing.Real`

Module of the spur gear.

property `SpurGear.pressure_angle`: `pint.Quantity`

Pressure angle of the spur gear.

property `SpurGear.n_teeth`: `int`

Number of teeth of the spur gear.

property `SpurGear.ref_diameter`: `ansys.geometry.core.typing.Real`

Reference diameter of the spur gear.

property `SpurGear.base_diameter`: `ansys.geometry.core.typing.Real`

Base diameter of the spur gear.

property `SpurGear.addendum`: `ansys.geometry.core.typing.Real`

Addendum of the spur gear.

property `SpurGear.dedendum`: `ansys.geometry.core.typing.Real`

Dedendum of the spur gear.

property `SpurGear.tip_diameter`: `ansys.geometry.core.typing.Real`

Tip diameter of the spur gear.

property `SpurGear.root_diameter`: `ansys.geometry.core.typing.Real`

Root diameter of the spur gear.

Description

Module for creating and managing gears.

The `polygon.py` module

Summary

Classes

<i>Polygon</i> Provides for modeling regular polygons.
--

Polygon

```
class ansys.geometry.core.sketch.polygon.Polygon(center: ansys.geometry.core.math.point.Point2D,
        inner_radius: beartype.typing.Union[pint.Quantity,
        ansys.geometry.core.misc.measurements.Distance,
        ansys.geometry.core.typing.Real], sides: int, angle:
        beartype.typing.Optional[beartype.typing.Union[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

[Union[Quantity, Distance, Real]] Inner radius (apothem) of the polygon.

sides

[int] Number of sides of the polygon.

angle

[Union[Quantity, Angle, Real], default: 0] Placement angle for orientation alignment.

Overview

Properties

<code>center</code>	Center point of the polygon.
<code>inner_radius</code>	Inner radius (apothem) of the polygon.
<code>n_sides</code>	Number of sides of the polygon.
<code>angle</code>	Orientation angle of the polygon.
<code>length</code>	Side length of the polygon.
<code>outer_radius</code>	Outer radius of the polygon.
<code>perimeter</code>	Perimeter of the polygon.
<code>area</code>	Area of the polygon.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.polygon import Polygon
```


Property detail

property Polygon.**center**: `ansys.geometry.core.math.point.Point2D`

Center point of the polygon.

property Polygon.**inner_radius**: `pint.Quantity`

Inner radius (apothem) of the polygon.

property Polygon.**n_sides**: `int`

Number of sides of the polygon.

property Polygon.**angle**: `pint.Quantity`

Orientation angle of the polygon.

property Polygon.**length**: `pint.Quantity`

Side length of the polygon.

property Polygon.**outer_radius**: `pint.Quantity`

Outer radius of the polygon.

property Polygon.**perimeter**: `pint.Quantity`

Perimeter of the polygon.

property Polygon.**area**: `pint.Quantity`

Area of the polygon.

property Polygon.**visualization_polydata**: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Description

Provides for creating and managing a polygon.

The `segment.py` module

Summary

Classes

<code>SketchSegment</code>	Provides segment representation of a line.
----------------------------	--

SketchSegment

```
class ansys.geometry.core.sketch.segment.SketchSegment(start:
    ansys.geometry.core.math.point.Point2D,
    end:
    ansys.geometry.core.math.point.Point2D,
    plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```

Bases: `ansys.geometry.core.sketch.edge.SketchEdge`, `ansys.geometry.core.shapes.curves.Line.Line`

Provides segment representation of a line.

Parameters

start

[Point2D] Starting point of the line segment.

end

[Point2D] Ending point of the line segment.

plane

[Plane, optional] Plane containing the sketched circle, which is the global XY plane by default.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing SketchSegment objects.
---------------------------	--

Properties

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

Special methods

<code>__eq__</code>	Equals operator for the SketchSegment class.
<code>__ne__</code>	Not equals operator for the SketchSegment class.

Import detail

```
from ansys.geometry.core.sketch.segment import SketchSegment
```

Property detail

property `SketchSegment.start`: `ansys.geometry.core.math.point.Point2D`

Starting point of the segment.

property `SketchSegment.end`: `ansys.geometry.core.math.point.Point2D`

Ending point of the segment.

property `SketchSegment.length`: `pint.Quantity`

Length of the segment.

property `SketchSegment.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

Method detail

`SketchSegment.__eq__(other: SketchSegment) → bool`

Equals operator for the `SketchSegment` class.

`SketchSegment.__ne__(other: SketchSegment) → bool`

Not equals operator for the `SketchSegment` class.

`SketchSegment.plane_change(plane: ansys.geometry.core.math.plane.Plane) → None`

Redefine the plane containing `SketchSegment` objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched segment.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing a segment.

The `sketch.py` module

Summary

Classes

<i>Sketch</i> Provides for building 2D sketch elements.

Attributes

<i>SketchObject</i> Type used to refer to both <i>SketchEdge</i> and <i>SketchFace</i> as possible values.
--

Sketch

```
class ansys.geometry.core.sketch.sketch.Sketch(plane:
    beartype.typing.Optional[ansys.geometry.core.math.plane.Plane]
    = Plane())
```

Provides for building 2D sketch elements.

Overview

Methods

<i>translate_sketch_plane</i>	Translate the origin location of the active sketch plane.
<i>translate_sketch_plane_by_offset</i>	Translate the origin location of the active sketch plane by offsets.
<i>translate_sketch_plane_by_distance</i>	Translate the origin location active sketch plane by distance.
<i>get</i>	Get a list of shapes with a given tag.
<i>face</i>	Add a sketch face to the sketch.
<i>edge</i>	Add a sketch edge to the sketch.
<i>select</i>	Add all objects that match provided tags to the current context.
<i>segment</i>	Add a segment sketch object to the sketch plane.
<i>segment_to_point</i>	Add a segment to the sketch plane starting from the previous edge end point.
<i>segment_from_point_and_vector</i>	Add a segment to the sketch starting from a given starting point.
<i>segment_from_vector</i>	Add a segment to the sketch starting from the end point of the previous edge.
<i>arc</i>	Add an arc to the sketch plane.
<i>arc_to_point</i>	Add an arc to the sketch starting from the end point of the previous edge.
<i>arc_from_three_points</i>	Add an arc to the sketch plane from three given points.
<i>triangle</i>	Add a triangle to the sketch using given vertex points.
<i>trapezoid</i>	Add a triangle to the sketch using given vertex points.
<i>circle</i>	Add a circle to the plane at a given center.
<i>box</i>	Create a box on the sketch.
<i>slot</i>	Create a slot on the sketch.
<i>ellipse</i>	Create an ellipse on the sketch.
<i>polygon</i>	Create a polygon on the sketch.
<i>dummy_gear</i>	Create a dummy gear on the sketch.
<i>spur_gear</i>	Create a spur gear on the sketch.
<i>tag</i>	Add a tag to the active selection of sketch objects.
<i>plot</i>	Plot all objects of the sketch to the scene.
<i>plot_selection</i>	Plot the current selection to the scene.
<i>sketch_polydata</i>	Get polydata configuration for all objects of the sketch to the scene.
<i>sketch_polydata_faces</i>	Get polydata configuration for all faces of the sketch to the scene.
<i>sketch_polydata_edges</i>	Get polydata configuration for all edges of the sketch to the scene.

Properties

<i>plane</i>	Sketch plane configuration.
<i>edges</i>	List of all independently sketched edges.
<i>faces</i>	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`: `beartype.typing.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`: `beartype.typing.List[ansys.geometry.core.sketch.face.SketchFace]`

List of all independently sketched faces.

Method detail

`Sketch.translate_sketch_plane`(*translation*: `ansys.geometry.core.math.vector.Vector3D`) → `Sketch`

Translate the origin location of the active sketch plane.

Parameters

translation

[`Vector3D`] Vector defining the translation. Meters is the expected unit.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.translate_sketch_plane_by_offset`(*x*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance]` = `Quantity(0, DEFAULT_UNITS.LENGTH)`, *y*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance]` = `Quantity(0, DEFAULT_UNITS.LENGTH)`, *z*: `beartype.typing.Union[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

[`Union[Quantity, Distance]`, default: `Quantity(0, DEFAULT_UNITS.LENGTH)`] Amount to translate the origin of the x-direction.

y

[`Union[Quantity, Distance]`, default: `Quantity(0, DEFAULT_UNITS.LENGTH)`] Amount to translate the origin of the y-direction.

z
 [Union[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: beartype.typing.Union[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
 [Union[Quantity, Distance]] Distance to translate the origin.

Returns

Sketch
 Revised sketch state ready for further sketch actions.

`Sketch.get(tag: str) → beartype.typing.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: beartype.typing.Optional[str] = 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: beartype.typing.Optional[str] = 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: beartype.typing.Optional[str] = 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: beartype.typing.Optional[str] = None) → Sketch`

Add a segment to the sketch plane starting from the previous edge 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: beartype.typing.Optional[str] = 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*: *beartype.typing.Optional[str] = None*)

Add a segment to the sketch starting from the end point of the previous edge.

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*: *beartype.typing.Optional[bool] = False*, *tag*: *beartype.typing.Optional[str] = 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*: `beartype.typing.Optional[bool] = False`, *tag*: `beartype.typing.Optional[str] = None`) → `Sketch`

Add an arc to the sketch starting from the end point of the previous edge.

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*: `beartype.typing.Optional[str] = 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.triangle`(*point1*: ansys.geometry.core.math.point.Point2D, *point2*: ansys.geometry.core.math.point.Point2D, *point3*: ansys.geometry.core.math.point.Point2D, *tag*: *beartype.typing.Optional[str] = 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`(*width*: *beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]*, *height*: *beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]*, *slant_angle*: *beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]*, *nonsymmetrical_slant_angle*: *beartype.typing.Optional[beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]] = None*, *center*: *beartype.typing.Optional[ansys.geometry.core.math.point.Point2D] = ZERO_POINT2D*, *angle*: *beartype.typing.Optional[beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]] = 0*, *tag*: *beartype.typing.Optional[str] = None*) → *Sketch*

Add a triangle to the sketch using given vertex points.

Parameters

width

[Union[Quantity, Distance, Real]] Width of the slot main body.

height

[Union[Quantity, Distance, Real]] Height of the slot.

slant_angle

[Union[Quantity, Angle, Real]] Angle for trapezoid generation.

nonsymmetrical_slant_angle

[Union[Quantity, Angle, Real], default: *None*] Asymmetrical slant angles on each side of the trapezoid. The default is *None*, in which case the trapezoid is symmetrical.

center

[Point2D, default: (0, 0)] Center point of the trapezoid.

angle

[Optional[Union[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.circle`(*center*: ansys.geometry.core.math.point.Point2D, *radius*: beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real], *tag*: beartype.typing.Optional[str] = None) → Sketch

Add a circle to the plane at a given center.

Parameters**center: Point2D**

Center point of the circle.

radius

[Union[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*: beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real], *height*: beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real], *angle*: beartype.typing.Optional[beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]] = 0, *tag*: beartype.typing.Optional[str] = None) → Sketch

Create a box on the sketch.

Parameters**center: Point2D**

Center point of the box.

width

[Union[Quantity, Distance, Real]] Width of the box.

height

[Union[Quantity, Distance, Real]] Height of the box.

angle

[Union[Quantity, 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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *height*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *angle*: `beartype.typing.Optional[beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]] = 0`, *tag*: `beartype.typing.Optional[str] = None`) → *Sketch*

Create a slot on the sketch.

Parameters

center: Point2D

Center point of the slot.

width

[Union[Quantity, Distance, Real]] Width of the slot.

height

[Union[Quantity, Distance, Real]] Height of the slot.

angle

[Union[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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *minor_radius*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *angle*: `beartype.typing.Optional[beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]] = 0`, *tag*: `beartype.typing.Optional[str] = None`) → *Sketch*

Create an ellipse on the sketch.

Parameters

center: Point2D

Center point of the ellipse.

major_radius

[Union[Quantity, Distance, Real]] Semi-major axis of the ellipse.

minor_radius

[Union[Quantity, Distance, Real]] Semi-minor axis of the ellipse.

angle

[Union[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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *sides*: `int`, *angle*: `beartype.typing.Optional[beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]] = 0`, *tag*: `beartype.typing.Optional[str] = None`) → *Sketch*

Create a polygon on the sketch.

Parameters

center: Point2D

Center point of the polygon.

inner_radius

[`Union[Quantity, Distance, Real]`] Inner radius (apothem) of the polygon.

sides

[`int`] Number of sides of the polygon.

angle

[`Union[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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *inner_radius*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Distance, ansys.geometry.core.typing.Real]`, *n_teeth*: `int`, *tag*: `beartype.typing.Optional[str] = None`) → *Sketch*

Create a dummy gear on the sketch.

Parameters

origin

[`Point2D`] Origin of the gear.

outer_radius

[`Union[Quantity, Distance, Real]`] Outer radius of the gear.

inner_radius

[`Union[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*: `beartype.typing.Union[pint.Quantity, ansys.geometry.core.misc.measurements.Angle, ansys.geometry.core.typing.Real]`, *n_teeth*: `int`, *tag*: `beartype.typing.Optional[str] = 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

[Union[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: beartype.typing.Optional[bool] = False, screenshot: beartype.typing.Optional[str] = None, use_frame: beartype.typing.Optional[bool] = None, selected_pd_objects: beartype.typing.List[pyvista.PolyData] = None, **plotting_options: beartype.typing.Optional[dict])`

Plot all objects of the sketch to the scene.

Parameters

view_2d

[bool, default: False] Whether to represent the plot in a 2D format.

screenshot

[str, optional] Path for saving a screenshot of the image that is being represented.

use_frame

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

**plotting_options

[dict, optional] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`Sketch.plot_selection(view_2d: beartype.typing.Optional[bool] = False, screenshot: beartype.typing.Optional[str] = None, use_frame: beartype.typing.Optional[bool] = None, **plotting_options: beartype.typing.Optional[dict])`

Plot the current selection to the scene.

Parameters

view_2d

[bool, default: `False`] Whether to represent the plot in a 2D format.

screenshot

[str, optional] Path for saving a screenshot of the image that is being represented.

use_frame

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

****plotting_options**

[dict, optional] Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`Sketch.sketch_polydata()` → `beartype.typing.List[pyvista.PolyData]`

Get polydata configuration for all objects of the sketch to the scene.

Returns

`List[PolyData]`

List of the polydata configuration for all edges and faces in the sketch.

`Sketch.sketch_polydata_faces()` → `beartype.typing.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()` → `beartype.typing.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 used to refer to both `SketchEdge` and `SketchFace` as possible values.

The slot.py module

Summary

Classes

<i>Slot</i>	Provides for modeling a 2D slot.
-------------	----------------------------------

Slot

```
class ansys.geometry.core.sketch.slot.Slot(center: ansys.geometry.core.math.point.Point2D, width:
    beartype.typing.Union[pint.Quantity,
    ansys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real], height:
    beartype.typing.Union[pint.Quantity,
    ansys.geometry.core.misc.measurements.Distance,
    ansys.geometry.core.typing.Real], angle:
    beartype.typing.Optional[beartype.typing.Union[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
[Union[Quantity, Distance, Real]] Width of the slot main body.

height
[Union[Quantity, Distance, Real]] Height of the slot.

angle
[Union[Quantity, Angle, Real], default: 0] Placement angle for orientation alignment.

Overview

Properties

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

Import detail

```
from ansys.geometry.core.sketch.slot import Slot
```

Property detail

property Slot.**center**: *ansys.geometry.core.math.point.Point2D*

Center of the slot.

property Slot.**width**: *pint.Quantity*

Width of the slot.

property Slot.**height**: *pint.Quantity*

Height of the slot.

property Slot.**perimeter**: *pint.Quantity*

Perimeter of the slot.

property Slot.**area**: *pint.Quantity*

Area of the slot.

property Slot.**visualization_polydata**: *pyvista.PolyData*

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData

VTK *pyvista.Polydata* configuration.

Description

Provides for creating and managing a slot.

The `trapezoid.py` module

Summary

Classes

<i>Trapezoid</i>	Provides for modeling a 2D trapezoid.
------------------	---------------------------------------

Trapezoid

```
class ansys.geometry.core.sketch.trapezoid.Trapezoid(width: beartype.typing.Union[pint.Quantity,
an-
sys.geometry.core.misc.measurements.Distance,
ansys.geometry.core.typing.Real], height:
beartype.typing.Union[pint.Quantity, an-
sys.geometry.core.misc.measurements.Distance,
ansys.geometry.core.typing.Real], slant_angle:
beartype.typing.Union[pint.Quantity, an-
sys.geometry.core.misc.measurements.Angle,
ansys.geometry.core.typing.Real],
nonsymmetrical_slant_angle:
beartype.typing.Optional[beartype.typing.Union[pint.Quantity,
an-
sys.geometry.core.misc.measurements.Angle,
ansys.geometry.core.typing.Real]] = None,
center:
beartype.typing.Optional[ansys.geometry.core.math.point.Point2D
= ZERO_POINT2D, angle:
beartype.typing.Optional[beartype.typing.Union[pint.Quantity,
an-
sys.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

width

[Union[Quantity, Distance, Real]] Width of the trapezoid.

height

[Union[Quantity, Distance, Real]] Height of the trapezoid.

slant_angle

[Union[Quantity, Angle, Real]] Angle for trapezoid generation.

nonsymmetrical_slant_angle

[Union[Quantity, Angle, Real], default: `None`] Asymmetrical slant angles on each side of the trapezoid. The default is `None`, in which case the trapezoid is symmetrical.

center: Point2D, default: ZERO_POINT2D

Center point of the trapezoid.

angle

[Union[Quantity, Angle, Real], default: 0] Placement angle for orientation alignment.

Notes

If a nonsymmetrical slant angle is defined, the slant angle is applied to the left-most angle, and the nonsymmetrical slant angle is applied to the right-most angle.

Overview

Properties

<i>center</i>	Center of the trapezoid.
<i>width</i>	Width of the trapezoid.
<i>height</i>	Height of the trapezoid.
<i>visualization_polydata</i>	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.**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

<i>Triangle</i>	Provides for modeling 2D triangles.
-----------------	-------------------------------------

Triangle

```
class ansys.geometry.core.sketch.triangle.Triangle(point1: ansys.geometry.core.math.point.Point2D,  
                                                    point2: ansys.geometry.core.math.point.Point2D,  
                                                    point3: ansys.geometry.core.math.point.Point2D)
```

Bases: *ansys.geometry.core.sketch.face.SketchFace*

Provides for modeling 2D triangles.

Parameters

point1: Point2D

Point that represents a triangle vertex.

point2: Point2D

Point that represents a triangle vertex.

point3: Point2D

Point that represents a triangle vertex.

Overview

Properties

<i>point1</i>	Triangle vertex 1.
<i>point2</i>	Triangle vertex 2.
<i>point3</i>	Triangle vertex 3.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.triangle import Triangle
```

Property detail

property Triangle.**point1**: *ansys.geometry.core.math.point.Point2D*

Triangle vertex 1.

property Triangle.**point2**: *ansys.geometry.core.math.point.Point2D*

Triangle vertex 2.

property Triangle.**point3**: *ansys.geometry.core.math.point.Point2D*

Triangle vertex 3.

property Triangle.**visualization_polydata**: *pyvista.PolyData*

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a triangle.

Description

PyAnsys Geometry sketch subpackage.

The tools package

Summary

Submodules

<i>measurement_tools</i>	Provides tools for measurement.
<i>problem_areas</i>	The problem area definition.
<i>repair_tool_message</i>	Module for repair tool message.
<i>repair_tools</i>	Provides tools for repairing bodies.

The measurement_tools.py module

Summary

Classes

<i>Gap</i>	Represents a gap between two bodies.
<i>MeasurementTools</i>	Measurement tools for PyAnsys Geometry.

Gap

```
class ansys.geometry.core.tools.measurement_tools.Gap(distance: an-
                                                    sys.geometry.core.misc.measurements.Distance)
```

Represents a gap between two bodies.

Parameters

distance

[Distance] Distance between two sides of the gap.

Overview

Properties

<i>distance</i> Returns the closest distance between two bodies.
--

Import detail

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

Property detail

```
property Gap.distance: ansys.geometry.core.misc.measurements.Distance
```

Returns the closest distance between two bodies.

MeasurementTools

```
class ansys.geometry.core.tools.measurement_tools.MeasurementTools(grpc_client: an-
                                                                    sys.geometry.core.connection.GrpcClient)
```

Measurement tools for PyAnsys Geometry.

Parameters

grpc_client

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

Overview

Methods

<i>min_distance_between_objects</i> Find the gap between two bodies.
--

Import detail

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

Method detail

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

Find the gap between two bodies.

Parameters

body1

[Body] First body to measure the gap.

body2

[Body] Second body to measure the gap.

Returns

Gap

Gap between two bodies.

Description

Provides tools for measurement.

The `problem_areas.py` module

Summary

Classes

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

ProblemArea

```
class ansys.geometry.core.tools.problem_areas.ProblemArea(id: str, grpc_client: an-
                                                         sys.geometry.core.connection.GrpcClient)
```

Represents problem areas.

Parameters

id

[str] Server-defined ID for the problem area.

grpc_client

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

Overview

Abstract methods

```
fix Fix problem area.
```

Properties

```
id The id of the problem area.
```

Import detail

```
from ansys.geometry.core.tools.problem_areas import ProblemArea
```

Property detail

```
property ProblemArea.id: str
```

The id of the problem area.

Method detail

```
abstract ProblemArea.fix()
```

Fix problem area.

DuplicateFaceProblemAreas

```
class ansys.geometry.core.tools.problem_areas.DuplicateFaceProblemAreas(id: str, grpc_client:
    an-
    sys.geometry.core.connection.GrpcClient,
    faces:
    beartype.typing.List[ansys.geometry.core
```

Bases: ProblemArea

Provides duplicate face problem area definition.

Represents a duplicate face problem area with unique identifier and associated faces.

Parameters

id

[str] Server-defined ID for the body.

grpc_client

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

faces

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

Overview

Methods

fix Fix the problem area.

Properties

faces The list of the edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import DuplicateFaceProblemAreas
```

Property detail

property DuplicateFaceProblemAreas.**faces**:
beartype.typing.List[ansys.geometry.core.designer.face.Face]

The list of the edges connected to this problem area.

Method detail

DuplicateFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage

Message containing created and/or modified bodies.

MissingFaceProblemAreas

class *ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas*(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: beartype.typing.List[ansys.geometry.core.d*

Bases: ProblemArea

Provides missing face problem area definition.

Parameters

id

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

grpc_client

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

edges

[*List[Edge]*] List of edges associated with the design.

Overview

Methods

<i>fix</i>	Fix the problem area.
------------	-----------------------

Properties

<i>edges</i>	The list of the edges connected to this problem area.
--------------	---

Import detail

```
from ansys.geometry.core.tools.problem_areas import MissingFaceProblemAreas
```

Property detail

property `MissingFaceProblemAreas.edges:`
beartype.typing.List`[ansys.geometry.core.designer.edge.Edge]`

The list of the edges connected to this problem area.

Method detail

`MissingFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: RepairToolMessage
 Message containing created and/or modified bodies.

InexactEdgeProblemAreas

class `ansys.geometry.core.tools.problem_areas.InexactEdgeProblemAreas`(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: beartype.typing.List[ansys.geometry.core.d*

Bases: `ProblemArea`

Represents an inexact edge problem area with unique identifier and associated edges.

Parameters

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

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

edges
`[List[Edge]]` List of edges associated with the design.

Overview

Methods

fix Fix the problem area.

Properties

<code>edges</code>	The list of the edges connected to this problem area.
--------------------	---

Import detail

```
from ansys.geometry.core.tools.problem_areas import InexactEdgeProblemAreas
```

Property detail

property `InexactEdgeProblemAreas.edges:`
beartype.typing.List`[ansys.geometry.core.designer.edge.Edge]`

The list of the edges connected to this problem area.

Method detail

`InexactEdgeProblemAreas.fix()` → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

ExtraEdgeProblemAreas

class `ansys.geometry.core.tools.problem_areas.ExtraEdgeProblemAreas`(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: beartype.typing.List[ansys.geometry.core.designer.edge.Edge]*)

Bases: `ProblemArea`

Represents a extra edge problem area with unique identifier and associated edges.

Parameters

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

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

edges
[`List[Edge]`] List of edges associated with the design.

Overview

Properties

edges The list of the ids of the edges connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import ExtraEdgeProblemAreas
```

Property detail

property `ExtraEdgeProblemAreas.edges:`
beartype.typing.List`[ansys.geometry.core.designer.edge.Edge]`

The list of the ids of the edges connected to this problem area.

SmallFaceProblemAreas

class `ansys.geometry.core.tools.problem_areas.SmallFaceProblemAreas`(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, faces: beartype.typing.List[ansys.geometry.core.designer.face.Face]*)

Bases: `ProblemArea`

Represents a small face problem area with unique identifier and associated faces.

Parameters

id

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

grpc_client

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

faces

`[List[Face]]` List of edges associated with the design.

Overview

Methods

fix Fix the problem area.

Properties

<code>faces</code>	The list of the ids of the edges connected to this problem area.
--------------------	--

Import detail

```
from ansys.geometry.core.tools.problem_areas import SmallFaceProblemAreas
```

Property detail

property `SmallFaceProblemAreas.faces:`
beartype.typing.List`[ansys.geometry.core.designer.face.Face]`

The list of the ids of the edges connected to this problem area.

Method detail

`SmallFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

SplitEdgeProblemAreas

class `ansys.geometry.core.tools.problem_areas.SplitEdgeProblemAreas`(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: beartype.typing.List[ansys.geometry.core.designer.face.Face]*)

Bases: `ProblemArea`

Represents a split edge problem area with unique identifier and associated edges.

Parameters

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

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

edges
[List[Edge]] List of edges associated with the design.

Overview

Methods

<i>fix</i>	Fix the problem area.
------------	-----------------------

Properties

<i>edges</i>	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:`
beartype.typing.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*: `beartype.typing.List`[`ansys.geometry.core.des`])

Bases: `ProblemArea`

Represents a stitch face problem area with unique identifier and associated faces.

Parameters

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

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

bodies

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

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>bodies</i> The list of the bodies connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import StitchFaceProblemAreas
```

Property detail

property `StitchFaceProblemAreas.bodies:`
beartype.typing.List[`ansys.geometry.core.designer.body.Body`]

The list of the bodies connected to this problem area.

Method detail

`StitchFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`
Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

Description

The problem area definition.

The repair_tool_message.py module

Summary

Classes

<code>RepairToolMessage</code>	Provides return message for the repair tool methods.
--------------------------------	--

RepairToolMessage

```
class ansys.geometry.core.tools.repair_tool_message.RepairToolMessage(success: bool,
                                                                    created_bodies:
                                                                    beartype.typing.List[str],
                                                                    modified_bodies:
                                                                    beartype.typing.List[str])
```

Provides return message for the repair tool methods.

Overview

Properties

<code>success</code>	The success of the repair operation.
<code>created_bodies</code>	The list of the created bodies after the repair operation.
<code>modified_bodies</code>	The list of the modified bodies after the repair operation.

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`: `beartype.typing.List[str]`

The list of the created bodies after the repair operation.

property `RepairToolMessage.modified_bodies`: `beartype.typing.List[str]`

The list of the modified bodies after the repair operation.

Description

Module for repair tool message.

The `repair_tools.py` module

Summary

Classes

<code>RepairTools</code>	Repair tools for PyAnsys Geometry.
--------------------------	------------------------------------

RepairTools

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

Repair tools for PyAnsys Geometry.

Overview

Methods

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

Import detail

```
from ansys.geometry.core.tools.repair_tools import RepairTools
```

Method detail

```
RepairTools.find_split_edges(bodies: beartype.typing.List[ansys.geometry.core.designer.body.Body], angle: ansys.geometry.core.typing.Real = 0.0, length: ansys.geometry.core.typing.Real = 0.0) → beartype.typing.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: beartype.typing.List[ansys.geometry.core.designer.body.Body]*) →
beartype.typing.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: beartype.typing.List[ansys.geometry.core.designer.body.Body]*) →
beartype.typing.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_duplicate_faces**(*bodies: beartype.typing.List[ansys.geometry.core.designer.body.Body]*)
→
beartype.typing.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: beartype.typing.List[ansys.geometry.core.designer.body.Body]) → beartype.typing.List[ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas]`

Find the missing faces.

This method find the missing face problem areas and returns a list of missing face problem areas objects.

Parameters

bodies

[List[Body]] List of bodies that missing faces are investigated on.

Returns

List[MissingFaceProblemAreas]

List of objects representing missing face problem areas.

`RepairTools.find_small_faces(bodies: beartype.typing.List[ansys.geometry.core.designer.body.Body]) → beartype.typing.List[ansys.geometry.core.tools.problem_areas.SmallFaceProblemAreas]`

Find the small face problem areas.

This method finds and returns a list of ids of small face problem areas objects.

Parameters

bodies

[List[Body]] List of bodies that small faces are investigated on.

Returns

List[SmallFaceProblemAreas]

List of objects representing small face problem areas.

`RepairTools.find_stitch_faces(bodies: beartype.typing.List[ansys.geometry.core.designer.body.Body]) → beartype.typing.List[ansys.geometry.core.tools.problem_areas.StitchFaceProblemAreas]`

Return the list of stitch face problem areas.

This method find the stitch face problem areas and returns a list of ids of stitch face problem areas objects.

Parameters

bodies

[List[Body]] List of bodies that stitchable faces are investigated on.

Returns

List[StitchFaceProblemAreas]

List of objects representing stitch face problem areas.

Description

Provides tools for repairing bodies.

Description

PyAnsys Geometry tools subpackage.

The `errors.py` module

Summary

Exceptions

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

Functions

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

Constants

<code>SIGINT_TRACKER</code>

`GeometryRuntimeError`

exception `ansys.geometry.core.errors.GeometryRuntimeError`

Bases: `RuntimeError`

Provides error message to raise when Geometry service passes a runtime error.

Import detail

```
from ansys.geometry.core.errors import GeometryRuntimeError
```

`GeometryExitedError`

exception `ansys.geometry.core.errors.GeometryExitedError` (`msg='Geometry service has exited.'`)

Bases: `RuntimeError`

Provides error message to raise when Geometry service has exited.

Parameters

msg

[`str`, default: "Geometry service has exited."] Message to raise.

Import detail

```
from ansys.geometry.core.errors import GeometryExitedError
```

Description

Provides PyAnsys Geometry-specific errors.

Module detail

`errors.handler(sig, frame)`

Pass signal to the custom interrupt handler.

`errors.protect_grpc(func)`

Capture gRPC exceptions and raise a more succinct error message.

This method captures the `KeyboardInterrupt` exception to avoid segfaulting the Geometry service.

While this works some of the time, it does not work all of the time. For some reason, gRPC still captures SIGINT.

`errors.SIGINT_TRACKER = []`

The `logger.py` module

Summary

Classes

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

Functions

<code>addfile_handler</code>	Add a file handler to the input.
<code>add_stdout_handler</code>	Add a stdout handler to the logger.

Attributes

<code>string_to_loglevel</code>

Constants

<code>LOG_LEVEL</code>
<code>FILE_NAME</code>
<code>DEBUG</code>
<code>INFO</code>
<code>WARN</code>
<code>ERROR</code>
<code>CRITICAL</code>
<code>STDOUT_MSG_FORMAT</code>
<code>FILE_MSG_FORMAT</code>
<code>DEFAULT_STDOUT_HEADER</code>
<code>DEFAULT_FILE_HEADER</code>
<code>NEW_SESSION_HEADER</code>
<code>LOG</code>

PyGeometryCustomAdapter

class `ansys.geometry.core.logger.PyGeometryCustomAdapter`(*logger*, *extra=None*)

Bases: `logging.LoggerAdapter`

Keeps the reference to the Geometry service instance name dynamic.

If you use the standard approach, which is supplying *extra* input to the logger, you must input Geometry service instances each time you do a log.

Using adapters, you only need to specify the Geometry service instance that you are referring to once.

Overview

Methods

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

Attributes

<code>level</code>
<code>file_handler</code>
<code>stdout_handler</code>

Import detail

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

Attribute detail

`PyGeometryCustomAdapter.level`

`PyGeometryCustomAdapter.file_handler`

`PyGeometryCustomAdapter.stdout_handler`

Method detail

`PyGeometryCustomAdapter.process(msg, kwargs)`

Process the logging message and keyword arguments passed in to a logging call to insert contextual information. You can either manipulate the message itself, the keyword args or both. Return the message and kwargs modified (or not) to suit your needs.

Normally, you'll only need to override this one method in a `LoggerAdapter` subclass for your specific needs.

`PyGeometryCustomAdapter.log_to_file(filename: str = FILE_NAME, level: int = LOG_LEVEL)`

Add a file handler to the logger.

Parameters

filename

[`str`, default: "pyansys-geometry.log"] Name of the file to write log messages to.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`PyGeometryCustomAdapter.log_to_stdout(level=LOG_LEVEL)`

Add a standard output handler to the logger.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`PyGeometryCustomAdapter.setLevel(level='DEBUG')`

Change the log level of the object and the attached handlers.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

PyGeometryPercentStyle

class `ansys.geometry.core.logger.PyGeometryPercentStyle`(*fmt*, *, *defaults=None*)

Bases: `logging.PercentStyle`

Provides a common messaging style for the `PyGeometryFormatter` class.

Import detail

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

PyGeometryFormatter

class `ansys.geometry.core.logger.PyGeometryFormatter`(*fmt=STDOUT_MSG_FORMAT*,
datefmt=None, *style='%*', *validate=True*,
defaults=None)

Bases: `logging.Formatter`

Provides a `Formatter` class for overwriting default format styles.

Import detail

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

InstanceFilter

class `ansys.geometry.core.logger.InstanceFilter`(*name=""*)

Bases: `logging.Filter`

Ensures that the `instance_name` record always exists.

Overview**Methods**

<i>filter</i>	Ensure that the <code>instance_name</code> attribute is always present.
---------------	---

Import detail

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

Method detail

`InstanceFilter.filter(record)`

Ensure that the `instance_name` attribute is always present.

Logger

```
class ansys.geometry.core.logger.Logger(level=logging.DEBUG, to_file=False, to_stdout=True,
                                         filename=FILE_NAME)
```

Provides the logger used for each PyAnsys Geometry session.

This class allows you to add handlers to the logger to output messages to a file or to the standard output (stdout).

Parameters

level

[`int`, default: 10] Logging level to filter the message severity allowed in the logger. The default is 10, in which case the `logging.DEBUG` level is used.

to_file

[`bool`, default: `False`] Whether to write log messages to a file.

to_stdout

[`bool`, default: `True`] Whether to write log messages to the standard output.

filename

[`str`, default: "pyansys-geometry.log"] Name of the file to write log 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

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

Attributes

<code>file_handler</code>
<code>std_out_handler</code>

Special methods

<code>__getitem__</code>	Overload the access method by item for the <code>Logger</code> class.
--------------------------	---

Import detail

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

Attribute detail

`Logger.file_handler`

`Logger.std_out_handler`

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: beartype.typing.Optional[str] = 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: beartype.typing.Optional[int] = None) → PyGeometryCustomAdapter`

Add a logger for a Geometry service instance.

The Geometry service instance logger is a logger with an adapter that adds contextual information such as the Geometry service instance name. This logger is returned, and you can use it to log events as a normal logger. It is stored in the `_instances` field.

Parameters

name

[str] Name for the new instance logger.

client_instance

[GrpcClient] Geometry service GrpcClient object, which should contain the `get_name` method.

level

[int, default: None] Level of logging.

Returns

PyGeometryCustomAdapter

Logger adapter customized to add Geometry service information to the logs. You can use this class to log events in the same way you would with the `Logger` class.

`Logger.__getitem__(key)`

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

`Logger.add_handling_uncaught_exceptions(logger)`

Redirect the output of an exception to a logger.

Parameters

logger

[`str`] Name of the logger.

Description

Provides a general framework for logging in PyAnsys Geometry.

This module is built on the [Logging facility for Python](#). It is not intended to replace the standard Python logging library but rather provide a way to interact between its `logging` class and PyAnsys Geometry.

The loggers used in this module include the name of the instance, which is intended to be unique. This name is printed in all active outputs and is used to track the different Geometry service instances.

Logger usage

Global logger

There is a global logger named `PyAnsys_Geometry_global` that is created when `ansys.geometry.core.__init__` is called. If you want to use this global logger, you must call it at the top of your module:

```
from ansys.geometry.core import LOG
```

You can rename this logger to avoid conflicts with other loggers (if any):

```
from ansys.geometry.core import LOG as logger
```

The default logging level of `LOG` is `ERROR`. You can change this level and output lower-level messages with this code:

```
LOG.logger.setLevel("DEBUG")
LOG.file_handler.setLevel("DEBUG") # If present.
LOG.stdout_handler.setLevel("DEBUG") # If present.
```

Alternatively, you can ensure that all the handlers are set to the input log level with this code:

```
LOG.setLevel("DEBUG")
```

This logger does not log to a file by default. If you want, you can add a file handler with this code:

```
import os

file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
LOG.log_to_file(file_path)
```

This also sets the logger to be redirected to this file. If you want to change the characteristics of this global logger from the beginning of the execution, you must edit the `__init__` file in the directory `ansys.geometry.core`.

To log using this logger, call the desired method as a normal logger with:

```
>>> import logging
>>> from ansys.geometry.core.logging import Logger
>>> LOG = Logger(level=logging.DEBUG, to_file=False, to_stdout=True)
>>> LOG.debug("This is LOG debug message.")

DEBUG - - <ipython-input-24-80df150fe31f> - <module> - This is LOG debug message.
```

Instance logger

Every time an instance of the *Modeler* class is created, a logger is created and stored in `LOG._instances`. This field is a dictionary where the key is the name of the created logger.

These instance loggers inherit the `PyAnsys_Geometry_global` output handlers and logging level unless otherwise specified. The way this logger works is very similar to the global logger. If you want to add a file handler, you can use the `log_to_file()` method. If you want to change the log level, you can use the `setLevel()` method.

Here is an example of how you can use this logger:

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler()
>>> modeler._log.info("This is a useful message")

INFO - GRPC_127.0.0.1:50056 - <...> - <module> - This is a useful message
```

Other loggers

You can create your own loggers using a Python logging library as you would do in any other script. There would be no conflicts between these loggers.

Module detail

`logger.addfile_handler(logger, filename=FILE_NAME, level=LOG_LEVEL, write_headers=False)`

Add a file handler to the input.

Parameters

logger

[`logging.Logger`] Logger to add the file handler to.

filename

[`str`, default: "pyansys-geometry.log"] Name of the output file.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

write_headers

[`bool`, default: `False`] Whether to write the headers to the file.

Returns

Logger

Logger or `logging.Logger` object.

`logger.add_stdout_handler(logger, level=LOG_LEVEL, write_headers=False)`

Add a stdout handler to the logger.

Parameters**logger**

[`logging.Logger`] Logger to add the file handler to.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

write_headers

[`bool`, default: `False`] Whether to write headers to the file.

Returns**Logger**

Logger or `logging.Logger` object.

`logger.LOG_LEVEL`

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

`logger.DEBUG`

`logger.INFO`

`logger.WARN`

`logger.ERROR`

`logger.CRITICAL`

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

`logger.FILE_MSG_FORMAT`

`logger.DEFAULT_STDOUT_HEADER = Multiline-String`

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

`logger.DEFAULT_FILE_HEADER`

`logger.NEW_SESSION_HEADER`

`logger.LOG`

`logger.string_to_loglevel`

The `modeler.py` module

Summary

Classes

<i>Modeler</i>	Provides for interacting with an open session of the Geometry service.
----------------	--

Modeler

```
class ansys.geometry.core.modeler.Modeler(host: str = DEFAULT_HOST, port:
    beartype.typing.Union[str, int] = DEFAULT_PORT, channel:
    beartype.typing.Optional[grpc.Channel] = None,
    remote_instance:
    beartype.typing.Optional[ansys.platform.instancemanagement.Instance]
    = None, docker_instance:
    beartype.typing.Optional[ansys.geometry.core.connection.docker_instance.Local
    = None, product_instance:
    beartype.typing.Optional[ansys.geometry.core.connection.product_instance.Product
    = None, timeout:
    beartype.typing.Optional[ansys.geometry.core.typing.Real] =
    120, logging_level: beartype.typing.Optional[int] =
    logging.INFO, logging_file:
    beartype.typing.Optional[beartype.typing.Union[pathlib.Path,
    str]] = None, backend_type:
    beartype.typing.Optional[ansys.geometry.core.connection.backend.BackendType
    = None)
```

Provides for interacting with an open session of the Geometry service.

Parameters

host

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

port

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

channel

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

remote_instance

[ansys.platform.instancemanagement.Instance, default: None] Corresponding remote instance when the Geometry service is launched using PyPIM. This instance is deleted when the `GrpcClient.close` method is called.

docker_instance

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

product_instance

[ProductInstance, default: None] Corresponding local product instance when the product (Discovery or SpaceClaim) is launched through the `launch_modeler_with_geometry_service()`,

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

timeout

[Real, default: 120] Time in seconds for trying to achieve the connection.

logging_level

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

logging_file

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

Overview**Methods**

<code>create_design</code>	Initialize a new design with the connected client.
<code>get_active_design</code>	Get the active design on the modeler object.
<code>read_existing_design</code>	Read the existing design on the service with the connected client.
<code>close</code>	Modeler method for easily accessing the client's close method.
<code>open_file</code>	Open a file.
<code>run_discovery_script_file</code>	Run a Discovery script file.

Properties

<code>client</code>	Modeler instance client.
<code>repair_tools</code>	Access to repair tools.
<code>measurement_tools</code>	Access to measurement tools.

Special methods

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

Import detail

```
from ansys.geometry.core.modeler import Modeler
```

Property detail

property `Modeler.client`: *ansys.geometry.core.connection.client.GrpcClient*

Modeler instance client.

property `Modeler.repair_tools`: *ansys.geometry.core.tools.repair_tools.RepairTools*

Access to repair tools.

property `Modeler.measurement_tools`:
ansys.geometry.core.tools.measurement_tools.MeasurementTools

Access to measurement tools.

Method detail

`Modeler.create_design(name: str) → ansys.geometry.core.designer.design.Design`

Initialize a new design with the connected client.

Parameters

name

[str] Name for the new design.

Returns

Design

Design object created on the server.

`Modeler.get_active_design(sync_with_backend: bool = True) → ansys.geometry.core.designer.design.Design`

Get the active design on the modeler object.

Parameters

sync_with_backend

[bool, default: True] Whether to sync the active design with the remote service. If set to False, the active design may be out-of-sync with the remote service. This is useful when the active design is known to be up-to-date.

Returns

Design

Design object already existing on the modeler.

`Modeler.read_existing_design() → ansys.geometry.core.designer.design.Design`

Read the existing design on the service with the connected client.

Returns

Design

Design object already existing on the server.

`Modeler.close() → None`

Modeler method for easily accessing the client's close method.

`Modeler.open_file(file_path: str, upload_to_server: bool = True, import_options: ansys.geometry.core.misc.options.ImportOptions = ImportOptions()) → ansys.geometry.core.designer.design.Design`

Open a file.

This method imports a design into the service. On Windows, .scdocx and HOOPS Exchange formats are supported. On Linux, only the .scdocx format is supported.

If the file is a shattered assembly with external references, the whole containing folder will need to be uploaded. Ensure proper folder structure in order to prevent the uploading of unnecessary files.

Parameters

file_path

[str] Path of the file to open. The extension of the file must be included.

upload_to_server

[bool] True if the service is running on a remote machine. If service is running on the local machine, set to False, as there is no reason to upload the file.

import_options

[ImportOptions] Import options that toggle certain features when opening a file.

Returns

Design

Newly imported design.

Modeler.__repr__() → str

Represent the modeler as a string.

Modeler.run_discovery_script_file(file_path: str, script_args: beartype.typing.Optional[beartype.typing.Dict[str, str]] = None, import_design=False) → beartype.typing.Tuple[beartype.typing.Dict[str, str], beartype.typing.Optional[ansys.geometry.core.designer.design.Design]]

Run a Discovery script file.

Note: If arguments are passed to the script, they must be in the form of a dictionary. On the server side, the script will receive the arguments as a dictionary of strings, under the variable name argsDict. For example, if the script is called with the arguments run_discovery_script_file(..., script_args = {"length": "20"}, ...), the script will receive the dictionary argsDict with the key-value pair {"length": "20"}.

Note: If an output is expected from the script, it will be returned as a dictionary of strings. The keys and values of the dictionary are the variables and their values that the script returns. However, it is necessary that the script creates a dictionary called result with the variables and their values that are expected to be returned. For example, if the script is expected to return the number of bodies in the design, the script should create a dictionary called result with the key-value pair {"numBodies": numBodies}, where numBodies is the number of bodies in the design.

The implied API version of the script should match the API version of the running Geometry Service. DMS API versions 23.2.1 and later are supported. DMS is a Windows-based modeling service that has been containerized to ease distribution, execution, and remotability operations.

Parameters

file_path

[str] Path of the file. The extension of the file must be included.

script_args

[Optional[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`.

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.

Description

Provides for interacting with the Geometry service.

The typing.py module**Summary****Attributes**

<i>Real</i>	Type used to refer to both integers and floats as possible values.
<i>RealSequence</i>	Type used to refer to <code>Real</code> types as a <code>Sequence</code> 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.

3.1.2 Description

PyAnsys Geometry is a Python wrapper for the Ansys Geometry service.

3.1.3 Module detail

`core.USE_TRAME: bool = False`

Global constant for checking whether to use `trame` for visualization.

`core.DISABLE_MULTIPLE_DESIGN_CHECK: bool = False`

Global constant for disabling the `ensure_design_is_active` check.

Only set this to false if you are sure you want to disable this check and you will ONLY be working with one design.

`core.DOCUMENTATION_BUILD: bool`

Global flag to set when building the documentation to use the proper PyVista Jupyter backend.

`core.__version__`

PyAnsys Geometry version.

EXAMPLES

These examples demonstrate the behavior and usage of PyAnsys Geometry.

4.1 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](#).

4.1.1 PyAnsys Geometry 101: Math

The `math` module is the foundation of PyAnsys Geometry. This module is built on top of [NumPy](#), one of the most renowned mathematical Python libraries.

This example shows some of the main PyAnsys Geometry math objects and demonstrates why they are important prior to doing more exciting things in PyAnsys Geometry.

Perform required imports

Perform the required imports.

```
[1]: import numpy as np

from ansys.geometry.core.math import Plane, Point2D, Point3D, Vector2D, Vector3D, ↵
↳ UnitVector3D
```


Create points and vectors

Everything starts with `Point` and `Vector` objects, which can each be defined in a 2D or 3D form. These objects inherit from NumPy's `ndarray`, providing them with enhanced functionalities. When creating these objects, you must remember to pass in the arguments as a list (that is, with brackets `[]`).

Create 2D and 3D point and vectors.

```
Point3D([x, y, z])
Point2D([x, y])

Vector3D([x, y, z])
Vector2D([x, y])
```

You can perform standard mathematical operations on points and vectors.

Perform some standard operations on vectors.

```
[2]: vec_1 = Vector3D([1,0,0]) # x-vector
      vec_2 = Vector3D([0,1,0]) # y-vector

      print("Sum of vectors [1, 0, 0] + [0, 1, 0]:")
      print(vec_1 + vec_2) # sum

      print("\nDot product of vectors [1, 0, 0] * [0, 1, 0]:")
      print(vec_1 * vec_2) # dot

      print("\nCross product of vectors [1, 0, 0] % [0, 1, 0]:")
      print(vec_1 % vec_2) # cross

Sum of vectors [1, 0, 0] + [0, 1, 0]:
[1 1 0]

Dot product of vectors [1, 0, 0] * [0, 1, 0]:
0

Cross product of vectors [1, 0, 0] % [0, 1, 0]:
[0 0 1]
```

Create a vector from two points.

```
[3]: p1 = Point3D([12.4, 532.3, 89])
      p2 = Point3D([-5.7, -67.4, 46.6])

      vec_3 = Vector3D.from_points(p1, p2)
      vec_3

[3]: Vector3D([ -18.1, -599.7, -42.4])
```

Normalize a vector to create a unit vector, which is also known as a *direction*.

```
[4]: print("Magnitude of vec_3:")
      print(vec_3.magnitude)

      print("\nNormalized vec_3:")
      print(vec_3.normalize())
```

(continues on next page)

(continued from previous page)

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

```
Magnitude of vec_3:
601.4694173438911
```

```
Normalized vec_3:
[-0.03009297 -0.99705818 -0.07049402]
```

```
New magnitude:
1.0
```

Use the `UnitVector` class to automatically normalize the input for the unit vector.

```
[5]: uv = UnitVector3D([1,1,1])
      uv
```

```
[5]: UnitVector3D([0.57735027, 0.57735027, 0.57735027])
```

Perform a few more mathematical operations on vectors.

```
[6]: v1 = Vector3D([1, 0, 0])
      v2 = Vector3D([0, 1, 0])

print("Vectors are perpendicular:")
print(v1.is_perpendicular_to(v2))

print("\nVectors are parallel:")
print(v1.is_parallel_to(v2))

print("\nVectors are opposite:")
print(v1.is_opposite(v2))

print("\nAngle between vectors:")
print(v1.get_angle_between(v2))
print(f"{np.pi / 2} == pi/2")
```

```
Vectors are perpendicular:
True
```

```
Vectors are parallel:
False
```

```
Vectors are opposite:
False
```

```
Angle between vectors:
1.5707963267948966 radian
1.5707963267948966 == pi/2
```

Create planes

Once you begin creating sketches and bodies, Plane objects become very important. A plane is defined by these items:

- An origin, which consists of a 3D point
- Two directions (`direction_x` and `direction_y`), which are both `UnitVector3DObjects`

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

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])
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.1.2 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}.")

# 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}")
```

(continues on next page)

(continued from previous page)

```

print(f"Y Coordinate: {point_a_km.y}")
print(f"Z Coordinate: {point_a_km.z}\n")

# These points, although they are in different units, can be added together.
res = point_a + point_a_km

print("==== res = point_a + point_a_km =====")
print(f"numpy.ndarray: {res}")
print(f"X Coordinate: {res.x}")
print(f"Y Coordinate: {res.y}")
print(f"Z Coordinate: {res.z}")

===== Point3D([1,2,4]) =====
Point3D is a numpy.ndarray in SI units: [1. 2. 4.].
However, request each of the coordinates individually...

X Coordinate: 1 meter
Y Coordinate: 2 meter
Z Coordinate: 4 meter

===== Point3D([1,2,4], unit=UNITS.km) =====
Point3D is a numpy.ndarray in SI units: [1000. 2000. 4000.].
However, request each of the coordinates individually...

X Coordinate: 1 kilometer
Y Coordinate: 2 kilometer
Z Coordinate: 4 kilometer

===== res = point_a + point_a_km =====
numpy.ndarray: [1001. 2002. 4004.]
X Coordinate: 1001.0 meter
Y Coordinate: 2002.0 meter
Z Coordinate: 4004.0 meter

```

Use default units

PyAnsys Geometry implements the concept of *default units*.

```
[4]: from ansys.geometry.core.misc import DEFAULT_UNITS
```

```

print("=== Default unit length ===")
print(DEFAULT_UNITS.LENGTH)

print("=== Default unit angle ===")
print(DEFAULT_UNITS.ANGLE)

=== Default unit length ===
meter
=== Default unit angle ===
radian

```

It is important to differentiate between *client-side* default units and *server-side* default units. You are able to control both of them.

Print the default server unit length.

```
[5]: print("=== Default server unit length ===")
print(DEFAULT_UNITS.SERVER_LENGTH)
```

```
=== Default server unit length ===
meter
```

Use default units.

```
[6]: from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import DEFAULT_UNITS

DEFAULT_UNITS.LENGTH = UNITS.mm

point_2d_default_units = Point2D([3, 4])
print("This is a Point2D with default units")
print(f"X Coordinate: {point_2d_default_units.x}")
print(f"Y Coordinate: {point_2d_default_units.y}")
print(f"numpy.ndarray value: {point_2d_default_units}")
```

```
# Revert back to original default units
DEFAULT_UNITS.LENGTH = UNITS.m
```

```
This is a Point2D with default units
X Coordinate: 3 millimeter
Y Coordinate: 4 millimeter
numpy.ndarray value: [0.003 0.004]
```

PyAnsys Geometry has certain auxiliary classes implemented that provide proper unit checking when assigning values. Although they are basically intended for internal use of the library, you can define them for use.

```
[7]: from ansys.geometry.core.misc import Angle, Distance
```

Start with Distance. The main difference between a Quantity object (that is, from `pint import Quantity`) and a Distance is that there is an active check on the units passed (in case they are not the default ones). Here are some examples.

```
[8]: radius = Distance(4)
print(f"The radius is {radius.value}.")

# Reassign the value of the distance
radius.value = 7 * UNITS.cm
print(f"After reassignment, the radius is {radius.value}.")

# Change the units if desired
radius.unit = UNITS.cm
print(f"After changing its units, the radius is {radius.value}.")
```

```
The radius is 4 meter.
After reassignment, the radius is 0.07 meter.
After changing its units, the radius is 7.0000000000000001 centimeter.
```

The next two code examples show how unreasonable operations raise errors.

```
[9]: try:
      radius.value = 3 * UNITS.degrees
    except TypeError as err:
      print(f"Error raised: {err}")
```

Error raised: The pint.Unit provided as an input should be a [length] quantity.

```
[10]: try:
       radius.unit = UNITS.fahrenheit
    except TypeError as err:
      print(f"Error raised: {err}")
```

Error raised: The pint.Unit provided as an input should be a [length] quantity.

The same behavior applies to the Angle object. Here are some examples.

```
[11]: import numpy as np

rotation_angle = Angle(np.pi / 2)
print(f"The rotation angle is {rotation_angle.value}.")

# Try reassigning the value of the distance
rotation_angle.value = 7 * UNITS.degrees
print(f"After reassignment, the rotation angle is {rotation_angle.value}.")

# You could also change its units if desired
rotation_angle.unit = UNITS.degrees
print(f"After changing its units, the rotation angle is {rotation_angle.value}.")
```

The rotation angle is 1.5707963267948966 radian.
After reassignment, the rotation angle is 0.12217304763960307 radian.
After changing its units, the rotation angle is 7.0 degree.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.1.3 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="Content-...
```

A *functional-style sketching API* is also implemented. It allows you to append curves to the sketch with the idea of *never picking up your pen*.

Use the functional-style sketching API to add a box.

```
[3]: sketch = Sketch()

(
    sketch.segment(Point2D([0,0]), Point2D([0,1]))
        .segment_to_point(Point2D([1,1]))
        .segment_to_point(Point2D([1,0]))
        .segment_to_point(Point2D([0,0]))
)

sketch.plot()

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

A Sketch object uses the XY plane by default. You can define your own custom plane using three parameters: `origin`, `direction_x`, and `direction_y`.

Add a box on a custom plane.

```
[4]: plane = Plane(origin=Point3D([0,0,0]), direction_x=Vector3D([1,2,-1]), direction_
↪y=Vector3D([1,0,1]))

sketch = Sketch(plane)

sketch.box(Point2D([0,0]), 1, 1)

sketch.plot()

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

Combine concepts to create powerful sketches

Combine these simple concepts to create powerful sketches.

```
[5]: # Complex Fluent API Sketch - PCB

sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↪"LowerEdge")
        .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↪mm), tag="SupportedCorner")
        .segment_to_point(Point2D([41.5, 5], unit=UNITS.mm))
        .arc_to_point(Point2D([43, 6.5], unit=UNITS.mm), Point2D([43, 5], unit=UNITS.mm),
↪True)
        .segment_to_point(Point2D([55, 6.5], unit=UNITS.mm))
        .arc_to_point(Point2D([56.5, 8], unit=UNITS.mm), Point2D([55, 8], unit=UNITS.mm))
        .segment_to_point(Point2D([56.5, 35], unit=UNITS.mm))
        .arc_to_point(Point2D([55, 36.5], unit=UNITS.mm), Point2D([55, 35], unit=UNITS.mm))
        .segment_to_point(Point2D([0, 36.5], unit=UNITS.mm))
        .segment_to_point(Point2D([0, 0], unit=UNITS.mm))
)
```

(continues on next page)

(continued from previous page)

```
.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

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.1.4 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)
```

```
Anslys Geometry Modeler (0x25a400adf70)
```

```
Anslys Geometry Modeler Client (0x25a400d7c20)
```

```
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)
)
```

(continues on next page)

(continued from previous page)

```
# 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")
```

(continues on next page)

(continued from previous page)

```

# Copy the body that belongs to this new component
body_copy = body.copy(parent=comp, name="Design_Component_Body")

# Displace this new body by a certain distance (10m) in a certain direction (X-axis)
body_copy.translate(direction=UNITVECTOR3D_X, distance=Distance(10, unit=UNITS.m))

# Plot the result of the entire design
design.plot()

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

```

Create and assign materials to the bodies that were created.

```

[6]: from pint import Quantity

from ansys.geometry.core.materials import Material, MaterialProperty,
↪MaterialPropertyType

# Define some general properties for the material.
density = Quantity(125, 10 * UNITS.kg / (UNITS.m * UNITS.m * UNITS.m))
poisson_ratio = Quantity(0.33, UNITS.dimensionless)
tensile_strength = Quantity(45) # WARNING: If no units are defined,
#it is assumed that the magnitude is in the units expected by the server.

# Once your material properties are defined, you can easily create a material.
material = Material(
    "steel",
    density,
    [MaterialProperty(MaterialPropertyType.POISSON_RATIO, "PoissonRatio", poisson_
↪ratio)],
)

# If you forgot to add a property, or you want to overwrite its value, you can still
# add properties to your created material.
material.add_property(
    type=MaterialPropertyType.TENSILE_STRENGTH, name="TensileProp", quantity=tensile_
↪strength
)

# Once your material is properly defined, send it to the server.
# This material can then be reused by different objects
design.add_material(material)

# Assign your material to your existing bodies.
body.assign_material(material)
body_copy.assign_material(material)

```

Currently materials do not have any impact on the visualization when plotting is requested, although this could be a future feature. If the final assembly is open in Discovery or SpaceClaim, you can observe the changes.

Create a named selection

PyAnsys Geometry supports the creation of a named selection via the Design object.

Create a named selection with some of the faces of the previous body and the body itself.

```
[7]: # Create a named selection
faces = body.faces
ns = design.create_named_selection("MyNamedSelection", bodies=[body], faces=[faces[0],
↪ faces[-1]])
print(f"This is a named selection called {ns.name} with ID (server-side): {ns.id}.")
```

```
This is a named selection called MyNamedSelection with ID (server-side): 0:429.
```

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 session instead. This is a safeguard for user-spawned services.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.1.5 PyAnsys Geometry 101: Plotter

This example provides an overview of PyAnsys Geometry’s plotting capabilities, focusing on its plotter features. After reviewing the fundamental concepts of sketching and modeling in PyAnsys Geometry, it shows how to leverage these key plotting capabilities:

- **Multi-object plotting:** You can conveniently plot a list of elements, including objects created in both PyAnsys Geometry and PyVista libraries.
- **Interactive object selection:** You can interactively select PyAnsys Geometry objects within the scene. This enables efficient manipulation of these objects in subsequent scripting.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity
import pyvista as pv

from ansys.geometry.core import Modeler
from ansys.geometry.core.connection.defaults import GEOMETRY_SERVICE_DOCKER_IMAGE
from ansys.geometry.core.connection.docker_instance import LocalDockInstance
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import PlotterHelper
from ansys.geometry.core.sketch import Sketch
```

Load modeling service

Load the modeling service. While the following code uses a Docker image to interact with the modeling service, you can use any suitable method mentioned in the preceding examples.

```
[2]: list_images = []
list_containers = []
available_images = LocalDockInstance.docker_client().images.list(
    name=GEOMETRY_SERVICE_DOCKER_IMAGE
)
is_image_available_cont = None
for image in available_images:
    for geom_image, geom_cont in zip(list_images, list_containers):
        if geom_image in image.tags:
            is_image_available = True
            is_image_available_cont = geom_cont
            break

docker_instance = LocalDockInstance(
    connect_to_existing_service=True,
    restart_if_existing_service=True,
    image=is_image_available_cont,
)

modeler = Modeler(docker_instance=docker_instance)
```

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 = PlotterHelper()
      plotter.plot(box_body2)
```

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

```
[9]: []
```

You can plot the whole list of objects.

```
[10]: plotter = PlotterHelper()
       plotter.plot(plot_list)
```

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

```
[10]: []
```

The Python visualizer is used by default. However, you can also use `trame` for visualization.

```
plotter = PlotterHelper(use_trame=True)
plotter.plot(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
      ph = PlotterHelper()

      # 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])
```

(continues on next page)

(continued from previous page)

```
# Add the object with the clipping plane
ph.add(box_body2, clipping_plane=plane)
ph.plot()

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

[11]: []

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 = PlotterHelper(allow_picking=True)

# Plotter returns picked bodies
picked_list = plotter.plot(plot_list)
print(picked_list)

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

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

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

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.2 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](#).

4.2.1 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
      from ansys.geometry.core.plotting import Plotter
```

Create a sketch

Sketches are fundamental objects for drawing basic shapes like lines, segments, circles, ellipses, arcs, and polygons.

You create a Sketch instance by defining a drawing plane. To define a plane, you declare a point and two fundamental orthogonal directions.

```
[2]: from ansys.geometry.core.math import Plane, Point2D, Point3D
```

Define a plane for creating a sketch.

```
[3]: # Define the origin point of the plane
      origin = Point3D([1, 1, 1])

      # Create a plane located in previous point with desired fundamental directions
      plane = Plane(
          origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
      )

      # Instantiate a new sketch object from previous plane
      sketch = Sketch(plane)
```

Draw shapes

To draw different shapes in the sketch, you use draw methods.

Draw a circle

You draw a circle in a sketch by specifying the center and radius.

```
[4]: sketch.circle(Point2D([2, 1]), radius=30 * u.cm, tag="Circle")
      sketch.select("Circle")
      sketch.plot_selection()
```

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

Draw an ellipse

You draw an ellipse in a sketch by specifying the center, major radius, and minor radius.

```
[5]: sketch.ellipse(
      Point2D([1, 1]), major_radius=2*u.m, minor_radius=1*u.m, tag="Ellipse"
    )
sketch.select("Ellipse")
sketch.plot_selection()

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

Draw a polygon

You draw a regular polygon by specifying the center, radius, and desired number of sides.

```
[6]: sketch.polygon(
      Point2D([1, 1]), inner_radius=3*u.m, sides=5, tag="Polygon"
    )
sketch.select("Polygon")
sketch.plot_selection()

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

Draw an arc

You draw an arc of circumference by specifying the center, starting point, and ending point.

```
[7]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
sketch.arc(start_point, end_point, Point2D([1,1]), tag="Arc")
sketch.select("Arc")
sketch.plot_selection()

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

Draw a slot

You draw a slot by specifying the center, width, and height.

```
[8]: 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.

```
[9]: 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.

```
[10]: 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()
```

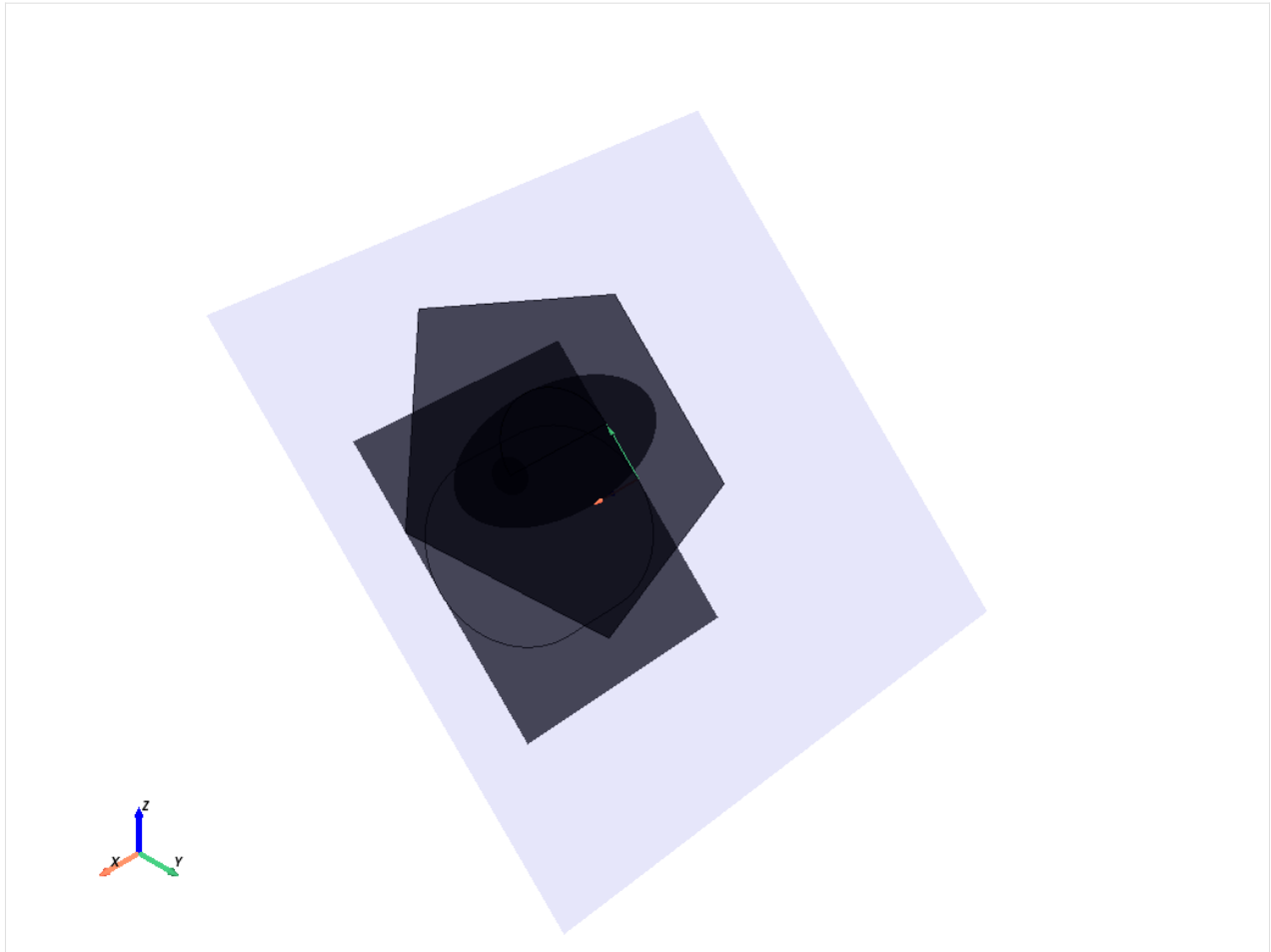
```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

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.

```
[11]: # Plot the sketch in the whole scene
      pl = Plotter()
      pl.plot_sketch(sketch, show_plane=True, show_frame=True)
      pl.scene.show(jupyter_backend="panel")
```



Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.2.2 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 Modeler
from ansys.geometry.core.math import UNITVECTOR3D_Z, Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile

You can create, modify, and plot `Sketch` instances independent of supporting Geometry service instances.

To define the sketch profile for the PCB, you create a sketch outline of individual `Segment` and `Arc` objects with two circular through-hole attachment points added within the profile boundary to maintain a single, closed sketch face.

Create a single `Sketch` instance to use for multiple design operations.

```
[2]: sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↳ "LowerEdge")
    .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↳ mm), tag="SupportedCorner")
    .segment_to_point(Point2D([41.5, 5], unit=UNITS.mm))
    .arc_to_point(Point2D([43, 6.5], unit=UNITS.mm), Point2D([43, 5], unit=UNITS.mm),
↳ True)
    .segment_to_point(Point2D([55, 6.5], unit=UNITS.mm))
    .arc_to_point(Point2D([56.5, 8], unit=UNITS.mm), Point2D([55, 8], unit=UNITS.mm))
    .segment_to_point(Point2D([56.5, 35], unit=UNITS.mm))
    .arc_to_point(Point2D([55, 36.5], unit=UNITS.mm), Point2D([55, 35], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 36.5], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 0], unit=UNITS.mm))
    .circle(Point2D([4, 4], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor1")
    .circle(Point2D([51, 34.5], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor2")
)

sketch.plot()

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

Extrude multiple bodies

Establish a server connection and 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).

```
[3]: modeler = Modeler()
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-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.2.3 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 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 Plotter
```

(continues on next page)

(continued from previous page)

```

# Start a modeler session
modeler = 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]
)

```

Sketch a dummy gear

DummyGear sketches are simple gears that have straight teeth. While they do not ensure actual physical functionality, they might be useful for some simple playground tests.

Instantiate a new Sketch object and then define and plot a dummy gear.

```

[2]: # Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

# Define dummy gear
#
origin = Point2D([0, 1], unit=UNITS.meter)
outer_radius = Distance(4, unit=UNITS.meter)
inner_radius = Distance(3.8, unit=UNITS.meter)
n_teeth = 30
sketch.dummy_gear(origin, outer_radius, inner_radius, n_teeth)

# Plot dummy gear
sketch.plot()

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

```

After creating the sketch, extrudes it.

```

[3]: # Create a design
design = modeler.create_design("AdvancedFeatures_DummyGear")

# Extrude your sketch
dummy_gear = design.extrude_sketch("DummyGear", sketch, Distance(1000, UNITS.mm))

# Plot the design
design.plot()

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

```

Sketch a spur gear

SpurGear sketches are parametric CAD spur gears based on four parameters:

- **origin**: Center point location for the desired spur gear. The value must be a `Point2D` object.
- **module**: Ratio between the pitch circle diameter in millimeters and the number of teeth. This is a common parameter for spur gears. The value should be an integer or a float.
- **pressure_angle**: Pressure angle expected for the teeth of the spur gear. This is also a common parameter for spur gears. The value must be a `pint.Quantity` object.
- **n_teeth**: Number of teeth. The value must be an integer.

Instantiate a new `Sketch` object and then define and plot a spur gear.

```
[4]: # Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

# Define spur gear
#
origin = Point2D([0, 1], unit=UNITS.meter)
module = 40
pressure_angle = Quantity(20, UNITS.deg)
n_teeth = 22

# Sketch spur gear
sketch.spur_gear(origin, module, pressure_angle, n_teeth)

# Plot spur gear
sketch.plot()
```

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

After creating the sketch, extrude it.

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

# 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="Content-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3 Modeling examples

These examples demonstrate service-based modeling operations.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.1 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 Modeler
from ansys.geometry.core.materials import Material, MaterialProperty,
↳MaterialPropertyType
from ansys.geometry.core.math import UNITVECTOR3D_Z, Frame, Plane, Point2D, Point3D,
↳UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Create sketch

Create a Sketch instance and insert a circle with a radius of 10 millimeters in the default plane.

```
[2]: sketch = Sketch()
sketch.circle(Point2D([10, 10], UNITS.mm), Quantity(10, UNITS.mm))

[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x1d52653b8f0>
```

Initiate design on server

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

```
[3]: modeler = Modeler()
      design_name = "ExtrudeProfile"
      design = modeler.create_design(design_name)
```

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))

      # 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-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.2 Modeling: Rectangular plate with multiple bodies

You can create multiple bodies from a single sketch by extruding the same sketch in different planes.

The sketch is designed as an effective *functional-style* API with all operations receiving 2D configurations. For more information, see the `:ref:Sketch` `<ref_sketch>` subpackage.

In this example, a box is located in the center of the plate, with the default origin of a sketch plane (origin at (0, 0, 0)). Four holes of equal radius are sketched at the corners of the plate. The plate is then extruded, leading to the generation of the requested body. The projection is at the center of the face. The default projection depth is through the entire part.

Perform required imports

Perform the required imports.

```
[1]: import numpy as np
      from pint import Quantity

      from ansys.geometry.core import Modeler
      from ansys.geometry.core.math import Plane, Point3D, Point2D, UnitVector3D
      from ansys.geometry.core.misc import UNITS
      from ansys.geometry.core.sketch import Sketch
```

Define sketch profile

The sketch profile for the proposed design requires four segments that constitute the outer limits of the design, a box on the center, and a circle at its four corners.

You can use a single `sketch` instance for multiple design operations, including extruding a body, projecting a profile, and imprinting curves.

Define the sketch profile for the rectangular plate with multiple bodies.

```
[2]: sketch = Sketch()
      (sketch.segment(Point2D([-4, 5], unit=UNITS.m), Point2D([4, 5], unit=UNITS.m))
        .segment_to_point(Point2D([4, -5], unit=UNITS.m))
        .segment_to_point(Point2D([-4, -5], unit=UNITS.m))
        .segment_to_point(Point2D([-4, 5], unit=UNITS.m))
        .box(Point2D([0,0], unit=UNITS.m), Quantity(3, UNITS.m), Quantity(3, UNITS.m))
        .circle(Point2D([3, 4], unit=UNITS.m), Quantity(0.5, UNITS.m))
        .circle(Point2D([-3, -4], unit=UNITS.m), Quantity(0.5, UNITS.m))
        .circle(Point2D([-3, 4], unit=UNITS.m), Quantity(0.5, UNITS.m))
        .circle(Point2D([3, -4], unit=UNITS.m), Quantity(0.5, UNITS.m))
      )
```

```
[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x1cfa00abd10>
```

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 = 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)
)

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="Content-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.3 Modeling: Tessellation of two bodies

This example shows how to create two stacked bodies and return the tessellation as two merged bodies.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import Modeler
from ansys.geometry.core.math import Point2D, Point3D, Plane
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import Plotter
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 = 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(merge_bodies=True)
dataset
```

```
[3]: MultiBlock (0x1ed3b25bac0)
    N Blocks      1
    X Bounds      -25.000, 25.000
    Y Bounds      -24.999, 34.999
    Z Bounds       0.000, 20.000
```

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

Code without merging the body

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

```
[4]: MultiBlock (0x1ed3b25bdc0)
    N Blocks      7
    X Bounds      -25.000, 25.000
    Y Bounds      -14.999, 34.999
    Z Bounds       10.000, 20.000
```

Code with merging the body

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

```
[5]: PolyData (0x1ed3b298520)
     N Cells:    1640
     N Points:   1650
     N Strips:    0
     X Bounds:  -2.500e+01, 2.500e+01
     Y Bounds:  -1.500e+01, 3.500e+01
     Z Bounds:   1.000e+01, 2.000e+01
     N Arrays:    0
```

Plot design

Plot the design.

```
[6]: design.plot()
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv=&quot;Content-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.4 Modeling: Design organization

The `Design` instance creates a design project within the remote Geometry service to complete all CAD modeling against.

You can organize all solid and surface bodies in each design within a customizable component hierarchy. A component is simply an organization mechanism.

The top-level design node and each child component node can have one or more bodies assigned and one or more components assigned.

The API requires each component of the design hierarchy to be given a user-defined name.

There are several design operations that result in a body being created within a design. Executing each of these methods against a specific component instance explicitly specifies the node of the design tree to place the new body under.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core import 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 = 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", UNITS.
↳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-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.5 Modeling: Boolean operations

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

Perform required imports

Perform the required imports.

```
[1]: from typing import List

from ansys.geometry.core import launch_docker_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 PlotterHelper
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_docker_modeler()
```

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.

```
bodies: List[Body] = PlotterHelper(allow_picking=True).plot(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)
_ = PlotterHelper().plot(design.bodies)

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

The final remaining body is the prism body because the cylin body has been consumed.

```
[8]: print(design.bodies)

[
ansys.geometry.core.designer.Body 0x1fba6607ad0
  Name          : Prism
  Exists        : True
  Parent component : intersection_design
  MasterBody    : 0:22
  Surface body   : False
]
```

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)
_ = PlotterHelper().plot(design.bodies)

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

The final remaining body is the prism body because the cylin body has been consumed.

```
[11]: print(design.bodies)

[
ansys.geometry.core.designer.Body 0x1fba6722de0
  Name           : Prism
  Exists         : True
  Parent component : union_design
  MasterBody     : 0:22
  Surface body   : False
]
```

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)
_ = PlotterHelper().plot(design.bodies)

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

The final remaining body is the prism body because the cylin body has been consumed.

```
[14]: print(design.bodies)

[
ansys.geometry.core.designer.Body 0x1fbad8797c0
  Name           : Prism
  Exists         : True
  Parent component : subtraction_design
  MasterBody     : 0:22
  Surface body   : False
]
```

If you perform this action inverting the order of the bodies (that is, `cylin.subtract(prism)`), you can see the difference in the resulting shape of the body.


```
[15]: # Create a design
design = modeler.create_design("subtraction_design_inverted")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

# Invert subtraction
cylin.subtract(prism)
_ = PlotterHelper().plot(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 0x1fbb5bd3fb0
  Name           : Cylinder
  Exists         : True
  Parent component : subtraction_design_inverted
  MasterBody     : 0:85
  Surface body   : False
]
```

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.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.6 Modeling: Scale, map and mirror bodies

The purpose of this notebook is to demonstrate the `map()` and `scale()` functions and their usage for transforming bodies.

```
[1]: # Imports
import numpy as np

from ansys.geometry.core import Modeler
from ansys.geometry.core.math import (
    Frame,
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.sketch import Sketch
```

```
[2]: # Initialize the modeler for this example notebook
m = Modeler()
```

Scale body

The `scale()` function is designed to modify the size of 3D bodies by a specified scale factor. This function is an important part of geometric transformations, allowing for the dynamic resizing of bodies.

Usage of `scale()`

To use the `scale()` function, you call it on an instance of a geometry body, passing a single argument: the scale value. This value is a real number (Real) that determines the factor by which the body's size will be changed.

```
body.scale(value)
```

Example: Making a cube

The following code snippets show how to change the size of a cube using the `scale()` function in `Body` objects. The process involves initializing a sketch design for the cube, defining the shape parameters, and then performing a rescaling operation to generate the new shape.

Initialize the cube sketch design

A new design sketch named “cube” is created.

```
[3]: design = m.create_design("cube")
```

Define cube parameters

side_length is set to 10 units, representing the side length of the cube.

```
[4]: # Cube parameters
side_length = 10
```

Create the profile cube

A square box is created centered on the origin using side_length as the side length of the square.

```
[5]: # Square with side length 10
box_sketch = Sketch().box(Point2D([0, 0]), side_length, side_length)

box_sketch.plot()

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

Create cube body

extrude_sketch on the box_sketch as the base sketch and create the 3D cube with distance being the side_length.

```
[6]: # Extrude the cube profile by a distance of side_length
cube = design.extrude_sketch("box", box_sketch, side_length)

design.plot()

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

Making the cube twice as large

- Copy the original cube. Using scale() with a value of 2, double the side lengths of the cube, thereby making the body twice as large, and then offset it to view the difference.

```
[7]: # Copy the original cube
doubled = cube.copy(cube.parent_component, "doubled_box")
# Double the size
doubled.scale(2)
# Translate the copied cube in the x direction
doubled.translate(UNITVECTOR3D_X, 30)
```

(continues on next page)

(continued from previous page)

```
design.plot()
```

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

Halving the size of the *original* cube

Copy the original cube. Using `scale()` with a value of 0.5 effectively halves the side lengths of the cube. Then, offset the new cube to view the difference.

Note: Because the size of the cube in the previous cell was doubled, using the 0.25 factor translates it to half the size of the original cube.

```
[8]: # Copy the original cube
halved = cube.copy(cube.parent_component, "halved_box")
# Half the size
halved.scale(0.5)
# Translate the copied cube in the x direction
halved.translate(UNITVECTOR3D_X, -25)
```

```
design.plot()
```

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

Map body

The `map()` function enables the reorientation of 3D bodies by mapping them onto a new specified frame. This function is used for adjusting the orientation of geometric bodies within 3D space to match specific reference frames. With this function, you are able to effectively perform translation and rotation operations in a single method by specifying a new frame.

Usage of `map()`

To use the `map()` function, invoke it on an instance of a geometry body with a single argument: the new frame to map the body to. The frame is a structure or object that defines the new orientation parameters for the body.

```
body.map(new_frame)
```

Example: Creating an asymmetric cube

The following code snippets show how to use the `map()` function to reframe a cube body in the `Body` object. The process involves initializing a sketch design for the custom body, extruding the profile by a distance, and then performing the mapping operation to rotate the shape.

Initialize the shape sketch design

A new design sketch named “asymmetric_cube” is created.

```
[9]: # Initialize the sketch design
design = m.create_design("asymmetric_cube")
```

Create an asymmetric sketch profile

Make a sketch profile that is basically a cube centered on the origin with a side length of 2 with a cutout.

```
[10]: # Create the cube profile with a cut through it
asymmetric_profile = Sketch()
(
    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="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="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))
```

(continues on next page)

(continued from previous page)

```
# 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="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="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="Content-...
```

Mirror the triangular body

We will first make a copy of the triangular body, then using `mirror()`, we will mirror the copied body over the ZY plane.

```
[17]: # Copy triangular body
mirrored_triangle = triangle.copy(triangle.parent_component, "mirrored_triangle")
# Mirror the copied body over the ZY plane (specified by the (0, 1, 0) and
# (0, 0, 1) unit vectors)

mirrored_triangle.mirror(Plane(direction_x=UNITVECTOR3D_Y, direction_y=UNITVECTOR3D_Z))

design.plot()

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

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.7 Modeling: Sweep chain and sweep sketch

This example shows how use the `sweep_sketch()` and `sweep_chain()` functions to create more complex extrusion profiles. You use the `sweep_sketch()` function with a closed sketch profile and the `sweep_chain()` function for an open profile.

```
[1]: # Imports
import numpy as np

from ansys.geometry.core import Modeler
from ansys.geometry.core.math import (
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.shapes import Circle, Ellipse, Interval
from ansys.geometry.core.sketch import Sketch

[2]: # Initialize the modeler for this example notebook
m = Modeler()
```

Example: Creating a donut

The following code snippets show how to use the `sweep_sketch()` function to create a 3D donut shape in the Design object. The process involves initializing a sketch design for the donut, defining two circles for the profile and path, and then performing a sweep operation to generate the donut shape.

Initialize the donut sketch design

A new design sketch named “donut” is created.

```
[3]: # Initialize the donut sketch design
design_sketch = m.create_design("donut")
```


Define circle parameters

`path_radius` is set to 5 units, representing the radius of the circular path that the profile circle sweeps along. `profile_radius` is set to 2 units, representing the radius of the profile circle that sweeps along the path to create the donut body.

```
[4]: # Donut parameters
path_radius = 5
profile_radius = 2
```

Create the profile circle

A circle is created on the XZ-plane centered at the coordinates (5, 0, 0) with a radius defined by `profile_radius`. This circle serves as the profile or cross-sectional shape of the donut.

```
[5]: # Create the circular profile on the XZ-plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(direction_x=UNITVECTOR3D_X, direction_y=UNITVECTOR3D_Z)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([path_radius, 0]), profile_radius)

profile.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv="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="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 2π radians. This path defines the sweeping trajectory for the profile to create the bowl.

```
[11]: # Create circle on the plane parallel to the XY-plane but moved up
# by 5 units with radius 10
path = [Circle(Point3D([0, 0, radius / 2]), radius).trim(Interval(0, 2 * np.pi))]
```

Perform the sweep operation

The bowl body is generated by sweeping the quarter-ellipse profile along the circular path. The sweep operation molds the profile shape along the path to form the stretched bowl. The result of this operation is stored in the variable `body`.

```
[12]: # Create the bowl body
body = design_chain.sweep_chain("bowl_sweep", path, profile)

design_chain.plot()

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

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.3.8 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 Modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, UNITVECTOR3D_Y, Point2D
from ansys.geometry.core.sketch import Sketch

# Instantiate the modeler - this connects to the service
modeler = Modeler()

# Create a design
design = modeler.create_design("ExportDesignExample")

# Create a sketch
sketch = Sketch()

# Create a simple rectangle
sketch.box(Point2D([0, 0]), 10, 5)
```

(continues on next page)

(continued from previous page)

```

# 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 C:\actions-runner\_work\pyansys-geometry\pyansys-geometry\doc\source\
↪examples\03_modeling\ExportDesignExample.scdocx

```

Export to a file in a certain directory

```

[3]: from pathlib import Path

# Define a downloads directory
download_dir = Path.cwd() / "downloads"

# Export the design to a file in a certain directory
file_location = design.export_to_scdocx(download_dir)
print(f"Design exported to {file_location}")

```

```

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

```

Export to SCDOCX format

```
[4]: # Export the design to a file in the requested directory
file_location = design.export_to_scdocx(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\actions-runner_work\pyansys-geometry\pyansys-geometry\doc\source\
↪examples\03_modeling\downloads\ExportDesignExample.scdocx
Does the file exist? True

Export to Parasolid text format

```
[5]: # Export the design to a file in the requested directory
file_location = design.export_to_parasolid_text(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\actions-runner_work\pyansys-geometry\pyansys-geometry\doc\source\
↪examples\03_modeling\downloads\ExportDesignExample.xmt_txt
Does the file exist? True

Export to Parasolid binary format

```
[6]: # Export the design to a file in the requested directory
file_location = design.export_to_parasolid_bin(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Design exported to C:\actions-runner_work\pyansys-geometry\pyansys-geometry\doc\source\
↪examples\03_modeling\downloads\ExportDesignExample.xmt_bin
Does the file exist? True

Export to STEP format

```
[7]: # Export the design to a file in the requested directory
file_location = design.export_to_step(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

```
Design exported to C:\actions-runner\_work\pyansys-geometry\pyansys-geometry\doc\source\
↳examples\03_modeling\downloads\ExportDesignExample.stp
Does the file exist? True
```

Export to IGES format

```
[8]: # Export the design to a file in the requested directory
file_location = design.export_to_iges(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")

Design exported to C:\actions-runner\_work\pyansys-geometry\pyansys-geometry\doc\source\
↳examples\03_modeling\downloads\ExportDesignExample.igs
Does the file exist? True
```

Export to FMD format

```
[9]: # Export the design to a file in the requested directory
file_location = design.export_to_fmd(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")

Design exported to C:\actions-runner\_work\pyansys-geometry\pyansys-geometry\doc\source\
↳examples\03_modeling\downloads\ExportDesignExample.fmd
Does the file exist? True
```

Export to PMDB format

```
[10]: # Export the design to a file in the requested directory
file_location = design.export_to_pmdb(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")

Design exported to C:\actions-runner\_work\pyansys-geometry\pyansys-geometry\doc\source\
↳examples\03_modeling\downloads\ExportDesignExample.pmdb
Does the file exist? True
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.4 Applied examples

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

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

4.4.1 Applied: Create a NACA 4-digit airfoil

NACA airfoils are a series of airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). They are a standardized system of airfoil shapes that are defined by a series of digits. The digits, which indicate the shape of the airfoil, are used to create the airfoil shape.

Each digit in the NACA airfoil number has a specific meaning:

- The first digit defines the maximum camber as a percentage of the chord length.
- The second digit defines the position of the maximum camber as a percentage of the chord length.
- The last two digits define the maximum thickness of the airfoil as a percentage of the chord length.

To fully understand the previous definitions, it is important to know that the chord length is the length of the airfoil from the leading edge to the trailing edge. The camber is the curvature of the airfoil, and the thickness is the distance between the upper and lower surfaces.

Symmetric airfoils have a camber of 0% and consequently, the first two digits of the NACA number are 0. For example, the NACA 0012 airfoil is a symmetric airfoil with a maximum thickness of 12%.

Define the NACA 4-digit airfoil equation

The following code uses the equation for a NACA 4-digit airfoil to create a set of points that define the airfoil shape. These points are `Point2D` objects in PyAnsys Geometry.

```
[1]: from typing import List, Union

import numpy as np

from ansys.geometry.core.math import Point2D

def naca_airfoil_4digits(number: Union[int, str], n_points: int = 200) -> List[Point2D]:
    """
    Generate a NACA 4-digits airfoil.

    Parameters
    -----
    number : int or str
        NACA 4-digit number.
    n_points : int
        Number of points to generate the airfoil. The default is ``200``.
        Number of points in the upper side of the airfoil.
        The total number of points is ``2 * n_points - 1``.
```

(continues on next page)

(continued from previous page)

```

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 or not
    if p == 0 and m == 0:
        # Camber line is zero in this case
        yc = 0
        dyc_dx = 0
    else:
        # Compute the camber line
        if x < p:
            yc = m / p**2 * (2 * p * x - x**2)
            dyc_dx = 2 * m / p**2 * (p - x)
        else:
            yc = m / (1 - p) ** 2 * ((1 - 2 * p) + 2 * p * x - x**2)
            dyc_dx = 2 * m / (1 - p) ** 2 * (p - x)

    # Compute the thickness
    yt = 5 * t * (0.2969 * x**0.5
                  - 0.1260 * x
                  - 0.3516 * x**2
                  + 0.2843 * x**3
                  - 0.1015 * x**4)

    # Compute the angle
    theta = np.arctan(dyc_dx)

    # Compute the points (upper and lower side of the airfoil)
    xu = x - yt * np.sin(theta)
    yu = yc + yt * np.cos(theta)
    xl = x + yt * np.sin(theta)
    yl = yc - yt * np.cos(theta)

```

(continues on next page)

(continued from previous page)

```
# 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="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="Content-...
```

Save the design

Finally, save the design to a file. This file can be used in other applications or imported into a simulation software. This code saves the design as an FMD file, which can then be imported into Ansys Fluent.

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

Design saved to C:\actions-runner\_work\pyansys-geometry\pyansys-geometry\doc\source\
↳examples\04_applied\NACA_Airfoil_0012.fmd
```

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.

```
[7]: 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.

```
[8]: # 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

```
[9]: # 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

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

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

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

CONTRIBUTE

Overall guidance on contributing to a PyAnsys library appears in the [Contributing](#) topic in the *PyAnsys Developer's Guide*. Ensure that you are thoroughly familiar with this guide before attempting to contribute to PyAnsys Geometry.

The following contribution information is specific to PyAnsys Geometry.

5.1 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 .
```

5.2 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.

5.3 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.

5.4 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"
black.....Passed
blacken-docs.....Passed
isort.....Passed
flake8.....Passed
docformatter.....Passed
codespell.....Passed
pydocstyle.....Passed
check for merge conflicts.....Passed
debug statements (python).....Passed
check yaml.....Passed
trim trailing whitespace.....Passed
Add License Headers.....Passed
Validate GitHub Workflows.....Passed
```

5.5 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
```

5.6 Run tests

PyAnsys Geometry uses `pytest` for testing.

5.6.1 Prerequisites

Prior to running the tests, you must run this command to install the test dependencies:

```
pip install -e .[tests]
```

Make sure to define the port and host of the service using the following environment variables:

```
# On Linux or macOS
export ANSRV_GEO_PORT=5000
export ANSRV_GEO_HOST=localhost

# On Windows CMD
set ANSRV_GEO_PORT=5000
set ANSRV_GEO_HOST=localhost
```

(continues on next page)

(continued from previous page)

```
# On Windows PowerShell
$env:ANSRV_GEO_PORT=5000
$env:ANSRV_GEO_HOST="localhost"
```

5.6.2 Running the tests

To run the tests, navigate to the root directory of the repository and run this command:

```
pytest
```

Note: The tests require the Geometry Service to be installed and running on your machine. The tests fail if the service is not running. It is expected for the Geometry Service to be running as a Docker container.

If you do not have the Geometry Service running as a Docker container, but you have it running on your machine, you can still run the tests with the following argument:

```
pytest --use-existing-service=yes
```

In this section, users are able to download a set of assets related to PyAnsys Geometry.

6.1 Documentation

The following links provide users with downloadable documentation in various formats

- [Documentation in HTML format](#)
- [Documentation in PDF format](#)

6.2 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 3.9 to 3.12. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

For example, on Linux with Python 3.9, unzip the wheelhouse archive and install it with:

```
unzip ansys-geometry-core-v0.5.3-wheelhouse-ubuntu-latest3.9.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

If you are on Windows with Python 3.9, 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:

6.2.1 Linux

- Linux wheelhouse for Python 3.9
- Linux wheelhouse for Python 3.10
- Linux wheelhouse for Python 3.11
- Linux wheelhouse for Python 3.12

6.2.2 Windows

- Windows wheelhouse for Python 3.9
- Windows wheelhouse for Python 3.10
- Windows wheelhouse for Python 3.11
- Windows wheelhouse for Python 3.12

6.2.3 MacOS

- MacOS wheelhouse for Python 3.9
- MacOS wheelhouse for Python 3.10
- MacOS wheelhouse for Python 3.11
- MacOS wheelhouse for Python 3.12

6.3 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.

6.3.1 Windows container

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

- Latest Geometry service binaries for Windows containers
- Latest Geometry service Dockerfile for Windows containers

RELEASE NOTES

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

7.1 0.5.3 - 2024-04-29

7.1.1 Fixed

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

7.2 0.5.2 - 2024-04-29

7.2.1 Added

- feat: add semver to intersphinx #1173

7.2.2 Changed

- chore: update CHANGELOG for v0.5.1 #1165
- chore: bump version to v0.6.dev0 #1166
- chore: update CHANGELOG for v0.5.2 #1172
- fix: allow to reuse last release binaries (if requested) #1174

7.2.3 Fixed

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

7.2.4 Miscellaneous

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

7.3 0.5.1 - 2024-04-24

7.3.1 Added

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

7.3.2 Changed

- ci: adapt to vale v3 #1129
- ci: bump ansys/actions from 5 to 6 in the actions group #1133
- docs: add release notes in our documentation #1138
- chore: bump ansys pre-commit hook to v0.3.0 #1139
- chore: use default vale version #1140
- docs: add `user_agent` to Sphinx build #1142
- ci: enabling Linux tests missing #1152
- ci: perform minimal requirements tests #1153

7.3.3 Fixed

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

7.3.4 Dependencies

- build: bump ansys-tools-path from 0.5.1 to 0.5.2 #1131
- build: bump the grpc-deps group across 1 directory with 3 updates #1156
- build: bump notebook from 7.1.2 to 7.1.3 in the docs-deps group #1157
- build: bump beartype from 0.18.2 to 0.18.5 #1158

7.3.5 Miscellaneous

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

7.4 0.5.0 - 2024-04-17

7.4.1 Added

- feat: inserting document into existing design #930
- feat: add changelog action #1023
- feat: create a sphere body on the backend #1035
- feat: mirror a body #1055
- feat: sweeping chains and profiles #1056
- feat: vulnerability checks #1071
- feat: loft profiles #1075
- feat: accept bandit advisories in-line for subprocess #1077
- feat: adding containers to automatic launcher #1090
- feat: minor changes to Linux Dockerfile #1111
- feat: avoid error if folder exists #1125

7.4.2 Changed

- build: changing sphinx-autoapi from 3.1.a2 to 3.1.a4 #1038
- chore: add pre-commit.ci configuration #1065
- chore: dependabot PR automatic approval #1067
- ci: bump the actions group with 1 update #1082
- chore: update docker tags to be kept #1085
- chore: update pre-commit versions #1094
- build: use ansys-sphinx-theme autoapi target #1097
- fix: removing @PipKat from *.md files - changelog fragments #1098
- ci: dashboard upload does not apply anymore #1099
- chore: pre-commit.ci not working properly #1108
- chore: update and adding pre-commit.ci config hook #1109
- ci: main Python version update to 3.12 #1112
- ci: skip Linux tests with common approach #1113
- ci: build changelog on release #1118

- chore: update CHANGELOG for v0.5.0 #1119

7.4.3 Fixed

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

7.4.4 Dependencies

- build: bump the docs-deps group with 2 updates #1062, #1093, #1105
- build: bump ansys-api-geometry from 0.3.13 to 0.4.0 #1066
- build: bump the docs-deps group with 1 update #1080
- build: bump pytest-cov from 4.1.0 to 5.0.0 #1081
- build: bump ansys-api-geometry from 0.4.0 to 0.4.1 #1092
- build: bump beartype from 0.17.2 to 0.18.2 #1106
- build: bump ansys-tools-path from 0.4.1 to 0.5.1 #1107
- build: bump panel from 1.4.0 to 1.4.1 in the docs-deps group #1114
- build: bump scipy from 1.12.0 to 1.13.0 #1115

7.4.5 Miscellaneous

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

PYTHON MODULE INDEX

a

ansys.geometry.core, ??
ansys.geometry.core.connection, ??
ansys.geometry.core.connection.backend, ??
ansys.geometry.core.connection.client, ??
ansys.geometry.core.connection.conversions, ??
ansys.geometry.core.connection.defaults, ??
ansys.geometry.core.connection.docker_instance, ??
ansys.geometry.core.connection.launcher, ??
ansys.geometry.core.connection.product_instance, ??
ansys.geometry.core.connection.validate, ??
ansys.geometry.core.designer, ??
ansys.geometry.core.designer.beam, ??
ansys.geometry.core.designer.body, ??
ansys.geometry.core.designer.component, ??
ansys.geometry.core.designer.coordinate_system, ??
ansys.geometry.core.designer.design, ??
ansys.geometry.core.designer.designpoint, ??
ansys.geometry.core.designer.edge, ??
ansys.geometry.core.designer.face, ??
ansys.geometry.core.designer.part, ??
ansys.geometry.core.designer.selection, ??
ansys.geometry.core.errors, ??
ansys.geometry.core.logger, ??
ansys.geometry.core.materials, ??
ansys.geometry.core.materials.material, ??
ansys.geometry.core.materials.property, ??
ansys.geometry.core.math, ??
ansys.geometry.core.math.bbox, ??
ansys.geometry.core.math.constants, ??
ansys.geometry.core.math.frame, ??
ansys.geometry.core.math.matrix, ??
ansys.geometry.core.math.plane, ??
ansys.geometry.core.math.point, ??
ansys.geometry.core.math.vector, ??
ansys.geometry.core.misc, ??
ansys.geometry.core.misc.accuracy, ??
ansys.geometry.core.misc.auxiliary, ??
ansys.geometry.core.misc.checks, ??
ansys.geometry.core.misc.measurements, ??
ansys.geometry.core.misc.options, ??
ansys.geometry.core.misc.units, ??
ansys.geometry.core.modeler, ??
ansys.geometry.core.plotting, ??
ansys.geometry.core.plotting.plotter, ??
ansys.geometry.core.plotting.plotter_helper, ??
ansys.geometry.core.plotting.plotting_types, ??
ansys.geometry.core.plotting.trame_gui, ??
ansys.geometry.core.plotting.widgets, ??
ansys.geometry.core.plotting.widgets.button, ??
ansys.geometry.core.plotting.widgets.displace_arrows, ??
ansys.geometry.core.plotting.widgets.measure, ??
ansys.geometry.core.plotting.widgets.ruler, ??
ansys.geometry.core.plotting.widgets.show_design_point, ??
ansys.geometry.core.plotting.widgets.view_button, ??
ansys.geometry.core.plotting.widgets.widget, ??
ansys.geometry.core.shapes, ??
ansys.geometry.core.shapes.box_uv, ??
ansys.geometry.core.shapes.curves, ??
ansys.geometry.core.shapes.curves.circle, ??
ansys.geometry.core.shapes.curves.curve, ??
ansys.geometry.core.shapes.curves.curve_evaluation, ??
ansys.geometry.core.shapes.curves.ellipse, ??
ansys.geometry.core.shapes.curves.line, ??
ansys.geometry.core.shapes.curves.trimmed_curve, ??
ansys.geometry.core.shapes.parameterization, ??
ansys.geometry.core.shapes-surfaces, ??
ansys.geometry.core.shapes-surfaces.cone, ??

ansys.geometry.core.shapes-surfaces.cylinder, ??
ansys.geometry.core.shapes-surfaces.plane, ??
ansys.geometry.core.shapes-surfaces.sphere, ??
ansys.geometry.core.shapes-surfaces.surface, ??
ansys.geometry.core.shapes-surfaces.surface_evaluation, ??
ansys.geometry.core.shapes-surfaces.torus, ??
ansys.geometry.core.shapes-surfaces.trimmed_surface, ??
ansys.geometry.core.sketch, ??
ansys.geometry.core.sketch.arc, ??
ansys.geometry.core.sketch.box, ??
ansys.geometry.core.sketch.circle, ??
ansys.geometry.core.sketch.edge, ??
ansys.geometry.core.sketch.ellipse, ??
ansys.geometry.core.sketch.face, ??
ansys.geometry.core.sketch.gears, ??
ansys.geometry.core.sketch.polygon, ??
ansys.geometry.core.sketch.segment, ??
ansys.geometry.core.sketch.sketch, ??
ansys.geometry.core.sketch.slot, ??
ansys.geometry.core.sketch.trapezoid, ??
ansys.geometry.core.sketch.triangle, ??
ansys.geometry.core.tools, ??
ansys.geometry.core.tools.measurement_tools, ??
ansys.geometry.core.tools.problem_areas, ??
ansys.geometry.core.tools.repair_tool_message, ??
ansys.geometry.core.tools.repair_tools, ??
ansys.geometry.core.typing, ??