



© 2026 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

Feb 19, 2026

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

CONTENTS

1	Installation	3
2	Available modes	5
3	Compatibility with Ansys releases	7
4	Development installation	9
5	Frequently asked questions	11
5.1	Docker containers	11
5.1.1	What is Docker?	11
5.1.2	Select your Docker container	11
5.2	Launch a local session	18
5.3	Launch a remote session	19
5.3.1	Set up the client machine	19
5.3.2	End the session	20
5.4	Use an existing session	20
5.4.1	Establish the connection	20
5.4.2	Verify the connection	21
5.5	Ansys version compatibility	21
5.6	Install package in development mode	21
5.6.1	Package dependencies	22
5.6.2	PyPI	22
5.6.3	Conda	22
5.6.4	GitHub	23
5.6.5	Install in offline mode	23
5.6.6	Verify your installation	23
5.7	Frequently asked questions	24
5.7.1	What is PyAnsys?	24
5.7.2	How is the Ansys Geometry Service installed?	24
5.7.3	What Ansys license is needed to run the Geometry service?	24
5.7.4	How to build the Docker image for the Ansys Geometry service?	25
6	User guide	27
6.1	Launching and connecting to a Modeler instance	27
6.1.1	Connecting to a Modeler instance	27
6.1.2	Connection types	28
6.1.3	Securing connections	28
6.2	Primitives	30
6.2.1	Planes	30
6.3	Sketch	31

6.3.1	Functional-style API	31
6.3.2	Direct API	31
6.4	Designer	32
6.4.1	Create the model	32
6.4.2	Define the model	32
6.4.3	Add materials to model	32
6.4.4	Create bodies by extruding the sketch	33
6.4.5	Create bodies by extruding the face	33
6.4.6	Download and save design	33
6.5	PyAnsys Geometry overview	33
6.6	Simple interactive example	34
6.6.1	Start Geometry server instance	34
6.6.2	Create geometry model	34
7	API reference	35
7.1	The <code>ansys.geometry.core</code> library	35
7.1.1	Summary	35
7.1.2	Description	453
7.1.3	Module detail	453
8	Examples	455
8.1	PyAnsys Geometry 101 examples	455
8.1.1	PyAnsys Geometry 101: Math	455
8.1.2	PyAnsys Geometry 101: Units	459
8.1.3	PyAnsys Geometry 101: Sketching	463
8.1.4	PyAnsys Geometry 101: Modeling	466
8.1.5	PyAnsys Geometry 101: Plotter	471
8.2	Sketching examples	476
8.2.1	Sketching: Basic usage	476
8.2.2	Sketching: Dynamic sketch plane	481
8.2.3	Sketching: Parametric sketching for gears	483
8.3	Modeling examples	487
8.3.1	Modeling: Single body with material assignment	487
8.3.2	Modeling: Rectangular plate with multiple bodies	489
8.3.3	Modeling: Extruded plate with cut operations	492
8.3.4	Modeling: Tessellation of two bodies	494
8.3.5	Modeling: Design organization	496
8.3.6	Modeling: Boolean operations	498
8.3.7	Modeling: Scale, map and mirror bodies	503
8.3.8	Modeling: Sweep chain and sweep sketch	509
8.3.9	Modeling: Revolving a sketch	513
8.3.10	Modeling: Exporting designs	516
8.3.11	Modeling: Visualization of the design tree on terminal	519
8.3.12	Modeling: Body color assignment and usage	525
8.3.13	Modeling: Surface bodies and trimmed surfaces	528
8.3.14	Modeling: Using design parameters	530
8.3.15	Modeling: Chamfer edges and faces	533
8.3.16	Modeling: Detach faces	535
8.4	Applied examples	541
8.4.1	Applied: Create a NACA 4-digit airfoil	542
8.4.2	Applied: Prepare a NACA airfoil for a Fluent simulation	547
8.4.3	Applied: Develop an external aerodynamic simulation model for Fluent analysis	550
8.5	Miscellaneous examples	559
8.5.1	Miscellaneous: Example template	559

9	Contribute	563
9.1	Clone the repository	563
9.2	Post issues	563
9.3	View documentation	563
9.4	Adhere to code style	564
9.5	Build the documentation	564
9.6	Adding examples	565
9.7	Run tests	566
9.7.1	Prerequisites	566
9.7.2	Running the tests	566
10	Assets	567
10.1	Documentation	567
10.2	Wheelhouse	567
10.2.1	Linux	567
10.2.2	Windows	568
10.2.3	MacOS	568
10.3	Geometry service Docker container assets	568
10.3.1	Windows container	568
11	Release notes	569
11.1	0.14.2 - January 07, 2026	569
11.2	0.14.1 - November 26, 2025	571
11.3	0.14.0 - November 26, 2025	571
11.4	0.13.0 - November 24, 2025	571
11.5	0.12.1 - October 21, 2025	573
11.6	0.12.0 - October 20, 2025	573
11.7	0.11.3 - September 17, 2025	576
11.8	0.11.2 - August 26, 2025	577
11.9	0.11.1 - August 13, 2025	577
11.10	0.11.0 - July 15, 2025	579
11.11	0.10.9 - June 05, 2025	580
11.12	0.10.8 - May 27, 2025	581
11.13	0.10.7 - May 05, 2025	582
11.14	0.10.6 - April 22, 2025	583
11.15	0.10.5 - April 16, 2025	583
11.16	0.10.4 - April 09, 2025	584
11.17	0.10.3 - April 08, 2025	585
11.18	0.10.2 - March 26, 2025	585
11.19	0.10.1 - March 21, 2025	586
11.20	0.10.0 - March 21, 2025	586
11.21	0.9.2 - April 16, 2025	587
11.21.1	Fixed	587
11.22	0.9.1 - 2025-02-28	588
11.22.1	Added	588
11.22.2	Dependencies	588
11.22.3	Fixed	588
11.22.4	Maintenance	588
11.23	0.9.0 - 2025-02-18	588
11.23.1	Added	588
11.23.2	Dependencies	589
11.23.3	Documentation	589
11.23.4	Fixed	589
11.23.5	Maintenance	589

11.23.6	Test	589
11.24	0.8.3 - April 16, 2025	590
11.24.1	Fixed	590
11.25	0.8.2 - 2025-01-29	590
11.25.1	Added	590
11.25.2	Dependencies	591
11.25.3	Fixed	591
11.25.4	Maintenance	591
11.25.5	Test	591
11.26	0.8.1 - 2025-01-15	592
11.26.1	Dependencies	592
11.26.2	Fixed	592
11.26.3	Maintenance	592
11.27	0.8.0 - 2025-01-15	592
11.27.1	Added	592
11.27.2	Dependencies	592
11.27.3	Documentation	593
11.27.4	Fixed	593
11.27.5	Maintenance	593
11.28	0.7.6 - 2024-11-19	594
11.28.1	Added	594
11.28.2	Dependencies	594
11.28.3	Documentation	594
11.28.4	Maintenance	594
11.29	0.7.5 - 2024-10-31	594
11.29.1	Added	594
11.29.2	Dependencies	595
11.29.3	Documentation	595
11.29.4	Fixed	595
11.29.5	Maintenance	595
11.30	0.7.4 - 2024-10-11	595
11.30.1	Dependencies	595
11.30.2	Fixed	596
11.30.3	Maintenance	596
11.31	0.7.3 - 2024-10-09	596
11.31.1	Added	596
11.31.2	Dependencies	596
11.31.3	Documentation	597
11.31.4	Fixed	597
11.31.5	Maintenance	597
11.32	0.7.2 - 2024-09-11	597
11.32.1	Added	597
11.32.2	Dependencies	597
11.32.3	Documentation	597
11.32.4	Fixed	597
11.32.5	Maintenance	598
11.33	0.7.1 - 2024-09-06	598
11.33.1	Added	598
11.33.2	Dependencies	598
11.33.3	Documentation	598
11.33.4	Fixed	598
11.33.5	Maintenance	598
11.34	0.7.0 - 2024-08-13	599
11.34.1	Added	599

11.34.2	Dependencies	599
11.34.3	Documentation	599
11.34.4	Fixed	599
11.34.5	Maintenance	599
11.34.6	Miscellaneous	599
11.35	0.6.6 - 2024-08-01	599
11.35.1	Added	599
11.35.2	Changed	599
11.35.3	Fixed	600
11.35.4	Dependencies	600
11.35.5	Miscellaneous	600
11.36	0.6.5 - 2024-07-02	600
11.36.1	Changed	600
11.36.2	Fixed	600
11.36.3	Dependencies	601
11.36.4	Miscellaneous	601
11.37	0.6.4 - 2024-06-24	601
11.37.1	Added	601
11.37.2	Changed	601
11.37.3	Fixed	601
11.37.4	Dependencies	601
11.38	0.6.3 - 2024-06-18	601
11.38.1	Changed	601
11.38.2	Fixed	601
11.38.3	Dependencies	601
11.38.4	Miscellaneous	602
11.39	0.6.2 - 2024-06-17	602
11.39.1	Added	602
11.39.2	Changed	602
11.39.3	Fixed	602
11.40	0.6.1 - 2024-06-12	602
11.40.1	Added	602
11.40.2	Changed	602
11.40.3	Fixed	602
11.40.4	Dependencies	602
11.40.5	Miscellaneous	602
11.41	0.6.0 - 2024-06-07	603
11.41.1	Added	603
11.41.2	Changed	603
11.41.3	Fixed	603
11.41.4	Dependencies	603
11.41.5	Miscellaneous	604
11.42	0.5.6 - 2024-05-23	604
11.42.1	Added	604
11.42.2	Changed	604
11.42.3	Dependencies	604
11.43	0.5.5 - 2024-05-21	604
11.43.1	Changed	604
11.43.2	Fixed	604
11.43.3	Dependencies	604
11.43.4	Miscellaneous	604
11.44	0.5.4 - 2024-05-15	604
11.44.1	Added	604
11.44.2	Changed	605

11.44.3	Dependencies	605
11.44.4	Miscellaneous	605
11.45	0.5.3 - 2024-04-29	605
11.45.1	Fixed	605
11.46	0.5.2 - 2024-04-29	605
11.46.1	Added	605
11.46.2	Changed	605
11.46.3	Fixed	605
11.46.4	Miscellaneous	605
11.47	0.5.1 - 2024-04-24	606
11.47.1	Added	606
11.47.2	Changed	606
11.47.3	Fixed	606
11.47.4	Dependencies	606
11.47.5	Miscellaneous	606
11.48	0.5.0 - 2024-04-17	606
11.48.1	Added	606
11.48.2	Changed	607
11.48.3	Fixed	607
11.48.4	Dependencies	608
11.48.5	Miscellaneous	608
Python Module Index		609
Index		611



PyAnsys Geometry

PyAnsys Geometry is a Python client library for the Ansys Geometry service. You are looking at the documentation for version 0.15.dev0.

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

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

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

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

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

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

Assets

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

INSTALLATION

You can use `pip` to install PyAnsys Geometry.

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


AVAILABLE MODES

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

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

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

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

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

Use an existing session

COMPATIBILITY WITH ANSYS RELEASES

PyAnsys Geometry continues to evolve as the Ansys products move forward. For more information, see [Ansys product version compatibility](#).

DEVELOPMENT INSTALLATION

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

FREQUENTLY ASKED QUESTIONS

Any questions? Refer to [Q&A](#) before submitting an issue.

5.1 Docker containers

5.1.1 What is Docker?

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

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

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

5.1.2 Select your Docker container

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

Select the kind of Docker container you want to build:

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

Windows Docker container

Go to Getting started

Windows Docker container

Contents

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

- *Build the Docker image from available binaries*
- *Launch the Geometry service*
 - * *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 XXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXXX > GH_TOKEN.txt
```

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

Powershell

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

Windows CMD

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

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

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

Build the Geometry service Windows container

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

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

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

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

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

There are two build modes:

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

Note

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

Prerequisites

- Ensure that Docker is installed in your machine. If you do not have Docker available, see [Docker for Windows containers](#).

Build from available Ansys installation

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

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

```
python build_docker_windows.py
```

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

```
docker images

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

Build the Docker image from available binaries

Prior to building your image, follow these steps:

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

Note

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

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

To build your image, follow these instructions:

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

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

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

```
docker images

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

Launch the Geometry service

There are methods for launching the Geometry service:

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

Environment variables

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

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

You can also specify other optional environment variables:

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

Here are some terms to keep in mind:

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

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

Using PyAnsys Geometry launcher

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

Linux/Mac

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

Powershell

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

Windows CMD

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

Warning

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

Manual launch

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

Geometry service launcher

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

Using PyAnsys Geometry launcher

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

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

modeler = launch_modeler()
```

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

Manual launch

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

Linux/Mac

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

Powershell

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

Windows CMD

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

Warning

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

Connect to the Geometry service

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

```
from ansys.geometry.core import Modeler

modeler = Modeler()
```

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

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

Warning

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

Environment variables

Linux/Mac

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

Powershell

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

Windows CMD

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


Modeler function

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

Go to Docker containers

Go to Getting started

5.2 Launch a local session

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

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

Discovery

```
from ansys.geometry.core import launch_modeler_with_discovery

modeler = launch_modeler_with_discovery()
```

SpaceClaim

```
from ansys.geometry.core import launch_modeler_with_spaceclaim

modeler = launch_modeler_with_spaceclaim()
```

Geometry service

```
from ansys.geometry.core import launch_modeler_with_geometry_service

modeler = launch_modeler_with_geometry_service()
```

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

Powershell

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

Windows CMD

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

Linux

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

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

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

Note

For information about transport modes and secure connections, see the *Securing connections* section in the User Guide.

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

5.3 Launch a remote session

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

Warning

This option is only available for Ansys employees.

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

5.3.1 Set up the client machine

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

```
from ansys.discovery.core import launch_modeler_with_pimlight_and_discovery

disco = launch_modeler_with_pimlight_and_discovery("241")
```

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

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

```
from ansys.discovery.core import launch_modeler_with_pimlight_and_spaceclaim

sc = launch_modeler_with_pimlight_and_spaceclaim("version")

from ansys.discovery.core import launch_modeler_with_pimlight_and_geometry_service
```

(continues on next page)

(continued from previous page)

```
geo = launch_modeler_with_pimlight_and_geometry_service("version")
```

Note

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

5.3.2 End the session

To end the session, run the corresponding command:

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

Go to [Getting started](#)

5.4 Use an existing session

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

Warning

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

5.4.1 Establish the connection

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

```
from ansys.geometry.core import Modeler

modeler = Modeler(host="localhost", port=50051, transport_mode="wnua") # On Windows
# or
modeler = Modeler(host="localhost", port=50051, transport_mode="uds") # On Linux
# or
modeler = Modeler(host="localhost", port=50051, transport_mode="insecure") # For any OS,
↪ using insecure gRPC
```

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.

Note

Starting from PyAnsys Geometry 0.14, the `transport_mode` parameter is required. For detailed information about available transport modes, secure connections, and compatibility with different Ansys releases, see [Securing](#)

connections.

5.4.2 Verify the connection

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

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

Go to Getting started

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

5.6 Install package in development mode

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

Contents

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

5.6.1 Package dependencies

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

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

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

5.6.2 PyPI

Before installing PyAnsys Geometry, to ensure that you have the latest version of [pip](#), run this command:

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

Then, to install PyAnsys Geometry, run this command:

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

5.6.3 Conda

You can also install PyAnsys Geometry using [conda](#). First, ensure that you have the latest version:

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

Then, to install PyAnsys Geometry, run this command:

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

5.6.4 GitHub

To install the latest release from the [PyAnsys Geometry repository](#) on GitHub, run these commands:

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

To verify your development installation, run this command:

```
tox
```

5.6.5 Install in offline mode

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

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

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

```
unzip ansys-geometry-core-v0.15.dev0-all-wheelhouse-ubuntu-3.10.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

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

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

5.6.6 Verify your installation

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

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

Ansys Geometry Modeler (0x205c5c17d90)

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

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

Go to [Getting started](#)

5.7 Frequently asked questions

5.7.1 What is PyAnsys?

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

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

5.7.2 How is the Ansys Geometry Service installed?

Note

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

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

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

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

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

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

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

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

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

Note

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

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

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

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

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

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

Note

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

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

Go to Getting started

USER GUIDE

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

6.1 Launching and connecting to a Modeler instance

PyAnsys Geometry provides the ability to launch and connect to a Modeler instance either locally or remotely. A Modeler instance is a running instance of a CAD service that the PyAnsys Geometry client can communicate with.

CAD services that are supported include:

- Ansys Geometry Service
- Ansys Discovery
- Ansys SpaceClaim

6.1.1 Connecting to a Modeler instance

To connect to a Modeler instance, you can use the `Modeler()` class, which provides methods to connect to the desired CAD service. You can specify the connection parameters such as the host, port, and authentication details.

```
from ansys.geometry.core import Modeler

# Connect to a local Modeler instance
modeler = Modeler(host="localhost", port=12345, transport_mode=...)
```

The `transport_mode` parameter can take several values depending on the type of connection you want to establish. You can refer to the [Securing connections](#) section for more details on the available transport modes.

Warning

Required `transport_mode` parameter

Starting from PyAnsys Geometry 0.14, the `transport_mode` parameter is required when using the `Modeler()` class. Only if the `channel` parameter is provided explicitly, the `transport_mode` can be omitted.

However, if you are using the `launch_modeler()` function to launch a local Modeler instance, the `transport_mode` parameter is optional and defaults to the appropriate mode based on the operating system and environment.

6.1.2 Connection types

PyAnsys Geometry supports different connection types to connect to a Modeler instance, which differ in the environment in which the transport takes place:

- **Local connection:** The Modeler instance is running on the same machine as the PyAnsys Geometry client. This is typically used for development and testing purposes.
- **Remote connection:** The service instance is running on a different machine, and the Modeler client connects to it over the network. This is useful for accessing Modeler instances hosted on remote servers or cloud environments.

6.1.3 Securing connections

When connecting to a remote Modeler instance, it is important to ensure that the connection is secure, not only to protect sensitive data but also to comply with organizational security policies. These secure connections can be established using various methods, such as:

Warning

Secure connection compatibility

Secure connections (mTLS, UDS, WNUA) are only available starting from Ansys release 24R2. However, some releases may require specific Service Packs to enable secure connection support. Starting from Ansys release 26R1, secure connections are available out-of-the-box without requiring additional Service Packs.

If you are using an Ansys release prior to 24R2, you must use insecure connections.

- **mTLS:** Mutual Transport Layer Security (mTLS) is a security protocol that ensures both the client and server authenticate each other using digital certificates. This provides a high level of security for the connection. PyAnsys Geometry supports mTLS connections to Modeler instances. In that case, you need to provide the necessary certificates and keys when establishing the connection. More information on the certificates needed can be seen in [Generating certificates for mTLS](#). Make sure that the names of the files are in line with the previous link. An example of how to set up an mTLS connection is shown below:

Note

mTLS is the default transport mode when connecting to remote Modeler instances.

```
from ansys.geometry.core import Modeler, launch_modeler

# OPTION 1: Connect to a Modeler instance using mTLS
modeler = Modeler(
    host="remote_host",
    port=12345,
    transport_mode="mtls",
    certs_dir="path/to/certs_directory",
)

# OPTION 2: Launch the Modeler instance locally using mTLS
modeler = launch_modeler(
    transport_mode="mtls",
    certs_dir="path/to/certs_directory",
)
```

- **UDS:** Unix Domain Sockets (UDS) provide a way to establish secure connections between processes on the same machine. UDS connections are faster and more secure than traditional network connections, as they do not require network protocols. PyAnsys Geometry supports UDS connections to local Modeler instances (**only on Linux-based services**). An example of how to set up a UDS connection is shown below:

Note

UDS is only supported when connecting to local Modeler instances on Linux-based services. It is also the default transport mode in such cases.

```
from ansys.geometry.core import Modeler, launch_modeler

# OPTION 1: Connect to a local Modeler instance using UDS
modeler = Modeler(host="localhost", port=12345, transport_mode="uds")

# OPTION 2: Launch the Modeler instance locally using UDS and specific directory,
# and id for
# the UDS socket
modeler = Modeler(host="localhost", port=12345, transport_mode="uds", uds_dir="/
↳path/to/uds_directory", uds_id="unique_id")

# OPTION 3: Launch the Modeler instance locally using UDS
modeler = launch_modeler(transport_mode="uds")

# OPTION 4: Launch the Modeler instance locally using UDS and specific directory,
# and id for
# the UDS socket
modeler = launch_modeler(transport_mode="uds", uds_dir="/path/to/uds_directory",
↳uds_id="unique_id")
```

- **WNUA:** Windows Named User Authentication (WNUA) provides a way to establish secure connections between processes on the same Windows machine. WNUA connections use a built-in mechanism to verify that the owner of the service running is also the owner of the client connection established (similar to UDS). PyAnsys Geometry supports WNUA connections to local Modeler instances (**only on Windows-based services**). An example of how to set up a WNUA connection is shown below:

Note

WNUA is only supported when connecting to local Modeler instances on Windows-based services. It is also the default transport mode in such cases.

```
from ansys.geometry.core import Modeler, launch_modeler

# OPTION 1: Connect to a local Modeler instance using WNUA
modeler = Modeler(host="localhost", port=12345, transport_mode="wnua")

# OPTION 2: Launch the Modeler instance locally using WNUA
modeler = launch_modeler(transport_mode="wnua")
```

- **Insecure:** Insecure connections do not provide any security measures to protect the data transmitted between the client and server. This mode is not recommended for production use, as it exposes the connection to potential

security risks. However, it can be useful for development and testing purposes in trusted environments. An example of how to set up an insecure connection is shown below:

```
from ansys.geometry.core import Modeler, launch_modeler

# OPTION 1: Connect to a Modeler instance using an insecure connection
modeler = Modeler(
    host="remote_host",
    port=12345,
    transport_mode="insecure",
)

# OPTION 2: Launch the Modeler instance locally using an insecure connection
modeler = launch_modeler(
    transport_mode="insecure",
)
```

For more information on secure connections and transport modes, see [Securing gRPC connections](#).

6.2 Primitives

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

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

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

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

- [Accuracy\(\)](#)
- [Measurements\(\)](#)

6.2.1 Planes

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

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

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

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

6.3 Sketch

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

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

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

This code shows how to initialize a sketch:

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

sketch = Sketch()
```

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

6.3.1 Functional-style API

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

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

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

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

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

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

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

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

6.3.2 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"
```

(continues on next page)

(continued from previous page)

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

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

For more information on sketch shapes, see the [Sketch\(\)](#) subpackage.

6.4 Designer

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

6.4.1 Create the model

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

```
from ansys.geometry.core import Modeler  
  
# Create the modeler object itself  
modeler = Modeler()
```

6.4.2 Define the model

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

```
from ansys.geometry.core.sketch import Sketch  
from ansys.geometry.core.math import Point2D  
from ansys.geometry.core.misc import UNITS  
from pint import Quantity  
  
# Create a sketch and draw a circle on the client  
sketch = Sketch()  
sketch.circle(Point2D([10, 10], UNITS.mm), Quantity(10, UNITS.mm))  
  
# Create your design on the server  
design_name = "ExtrudeProfile"  
design = modeler.create_design(design_name)
```

6.4.3 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))
```

(continues on next page)

(continued from previous page)

```

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

```

6.4.4 Create bodies by extruding the sketch

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

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

6.4.5 Create bodies by extruding the face

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

```

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

```

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

- [`Body\(\)`](#)
- [`Component\(\)`](#)

6.4.6 Download and save design

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

```

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

```

For more information, see the [*Design*](#) submodule.

6.5 PyAnsys Geometry overview

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

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

6.6 Simple interactive example

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

6.6.1 Start Geometry server instance

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

This code starts an instance of the Geometry service:

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

6.6.2 Create geometry model

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

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

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

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

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

API REFERENCE

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

7.1 The `ansys.geometry.core` library

7.1.1 Summary

Subpackages

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

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

Constants

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

The connection package

Summary

Submodules

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

The backend.py module

Summary

Enums

<i>BackendType</i>	Provides an enum holding the available backend types.
<i>ApiVersions</i>	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

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

Static methods

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

Import detail

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

Attribute detail

`BackendType.DISCOVERY = 0`

`BackendType.SPACECLAIM = 1`

`BackendType.WINDOWS_SERVICE = 2`

`BackendType.LINUX_SERVICE = 3`

`BackendType.CORE_WINDOWS = 4`

`BackendType.CORE_LINUX = 5`

`BackendType.DISCOVERY_HEADLESS = 6`

Method detail

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

Determine whether the backend is CoreService based or not.

Parameters

backend_type

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

Returns

bool

True if the backend is CoreService based, False otherwise.

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

Determine whether the backend is a headless service or not.

Parameters

backend_type

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

Returns

bool

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

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

Determine whether the backend is a Linux service or not.

Parameters

backend_type

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

Returns

bool

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

ApiVersions

class ansys.geometry.core.connection.backend.**ApiVersions**(*args, **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>
<code>V_242</code>
<code>V_251</code>
<code>V_252</code>
<code>V_261</code>

Static methods

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

Import detail

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

Attribute detail

`ApiVersions.V_21 = 21`

`ApiVersions.V_22 = 22`

`ApiVersions.V_231 = 231`

`ApiVersions.V_232 = 232`

`ApiVersions.V_241 = 241`

`ApiVersions.V_242 = 242`

`ApiVersions.V_251 = 251`

`ApiVersions.V_252 = 252`

`ApiVersions.V_261 = 261`

Method detail

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

Convert an input to an `ApiVersions` enum.

Parameters

version

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

Returns

ApiVersions

The version as an `ApiVersions` enum.

Description

Module providing definitions for the backend types.

The `client.py` module

Summary

Classes

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

Functions

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

`GrpcClient`

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

Wraps the gRPC connection for the Geometry service.

Parameters

host

[[str](#), default: `DEFAULT_HOST`] Host where the server is running.

port

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

channel

[[Channel](#), default: `None`] gRPC channel for server communication.

remote_instance

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

docker_instance

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

product_instance

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

timeout

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

logging_level

[[int](#), default: `INFO`] Logging level to apply to the client.

logging_file

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

proto_version

[[str](#) | [None](#), default: [None](#)] Protocol version to use for communication with the server. If [None](#), v0 is used. Available versions are “v0”, “v1”, etc.

transport_mode

[[str](#) | [None](#)] Transport mode selected. Needed if `channel` is not provided. Options are: “insecure”, “uds”, “wnua”, “mtls”.

uds_dir

[Path | [str](#) | [None](#)] Directory to use for Unix Domain Sockets (UDS) transport mode. By default [None](#) and thus it will use the “~/conn” folder.

uds_id

[[str](#) | [None](#)] Optional ID to use for the UDS socket filename. By default [None](#) and thus it will use “apodas_socket.sock”. Otherwise, the socket filename will be “apodas_socket-<uds_id>.sock”.

certs_dir

[Path | [str](#) | [None](#)] Directory to use for TLS certificates. By default [None](#) and thus search for the “ANSYS_GRPC_CERTIFICATES” environment variable. If not found, it will use the “certs” folder assuming it is in the current working directory.

Overview

Methods

backend_info	Get a string with the backend information.
close	Close the channel.
target	Get the target of the channel.
get_name	Get the target name of the connection.

Properties

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

Special methods

__repr__	Represent the client as a string.
--------------------------	-----------------------------------

Import detail

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


Property detail

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

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

Notes

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

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

Get the current backend version.

Returns

Version

Backend version.

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

Client gRPC channel.

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

GRPC services.

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

Specific instance logger.

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

Flag indicating whether the client connection is closed.

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

Flag indicating whether the client channel is healthy.

Method detail

GrpcClient.**backend_info**(*indent=0*) → *str*

Get a string with the backend information.

Returns

str

String with the backend information.

GrpcClient.**__repr__**() → *str*

Represent the client as a string.

GrpcClient.**close**()

Close the channel.

Notes

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

GrpcClient.**target**() → *str*

Get the target of the channel.

GrpcClient.**get_name**() → *str*

Get the target name of the connection.

Description

Module providing a wrapped abstraction of the gRPC stubs.

Module detail

`ansys.geometry.core.connection.client.wait_until_healthy(channel: grpc.Channel | str, timeout: float, transport_mode: str | None = None, uds_dir: pathlib.Path | str | None = None, uds_id: str | None = None, certs_dir: pathlib.Path | str | None = None) → grpc.Channel`

Wait until a channel is healthy before returning.

Parameters

channel

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

timeout

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

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

transport_mode

[*str* | *None*] Transport mode selected. Needed if channel is a string. Options are: “insecure”, “uds”, “wnua”, “mtls”.

uds_dir

[*Path* | *str* | *None*] Directory to use for Unix Domain Sockets (UDS) transport mode. By default *None* and thus it will use the “~/conn” folder.

uds_id

[*str* | *None*] Optional ID to use for the UDS socket filename. By default *None* and thus it will use “aposdas_socket.sock”. Otherwise, the socket filename will be “aposdas_socket-<uds_id>.sock”.

certs_dir

[*Path* | *str* | *None*] Directory to use for TLS certificates. By default *None* and thus search for the “ANSYS_GRPC_CERTIFICATES” environment variable. If not found, it will use the “certs” folder assuming it is in the current working directory.

Returns

grpc.Channel

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

Raises

TimeoutError

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

The 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

`ansys.geometry.core.connection.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.

`ansys.geometry.core.connection.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.

`ansys.geometry.core.connection.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.

`ansys.geometry.core.connection.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.

`ansys.geometry.core.connection.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

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

Enums

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

Functions

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

LocalDockerInstance

```
class ansys.geometry.core.connection.docker_instance.LocalDockerInstance(port: int = py-
                                                                    geom_defaults.DEFAULT_PORT,
                                                                    con-
                                                                    nect_to_existing_service:
                                                                    bool = True,
                                                                    restart_if_existing_service:
                                                                    bool = False, name:
                                                                    str | None = None,
                                                                    image:
                                                                    GeometryContainers
                                                                    | None = None,
                                                                    transport_mode: str |
                                                                    None = None,
                                                                    certs_dir:
                                                                    pathlib.Path | str |
                                                                    None = None)
```

Instantiates a Geometry service as a local Docker container.

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

Parameters

port

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

connect_to_existing_service

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

restart_if_existing_service

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

name

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

image

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

transport_mode

[`str` | `None`] Transport mode selected, by default *None* and thus it will be selected for you based on the connection criteria. Options are: “insecure”, “mtls”

certs_dir

[Path | `str` | `None`] Directory to use for TLS certificates. By default *None* and thus search for the “ANSYS_GRPC_CERTIFICATES” environment variable. If not found, it will use the “certs” folder assuming it is in the current working directory.

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.
<code>transport_mode</code>	Transport mode used by the Docker container.

Attributes

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

Static methods

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

Import detail

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

Property detail

property LocalDockeInstance.`container`: `docker.models.containers.Container`

Docker container object that hosts the deployed Geometry service.

property LocalDockeInstance.`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.

property LocalDockeInstance.`transport_mode`: `str`

Transport mode used by the Docker container.

Returns

`str`

Transport mode used by the Docker container. If no transport mode was specified during initialization, it defaults to “mtls”.

Attribute detail

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

Docker client class variable.

Notes

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

Method detail

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

Get the initialized `__DOCKER_CLIENT__` object.

Returns

`DockerClient`

Initialized Docker client.

Notes

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

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

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

Returns

`bool`

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

GeometryContainers

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

Bases: `enum.Enum`

Provides an enum holding the available Geometry services.

Overview

Attributes

<code>CORE_WINDOWS_LATEST</code>
<code>CORE_LINUX_LATEST</code>
<code>CORE_WINDOWS_LATEST_UNSTABLE</code>
<code>CORE_LINUX_LATEST_UNSTABLE</code>
<code>WINDOWS_LATEST</code>
<code>WINDOWS_LATEST_UNSTABLE</code>
<code>WINDOWS_24_1</code>
<code>WINDOWS_24_2</code>
<code>WINDOWS_25_1</code>
<code>WINDOWS_25_2</code>
<code>CORE_WINDOWS_25_2</code>
<code>CORE_LINUX_25_2</code>
<code>CORE_WINDOWS_26_1</code>
<code>CORE_LINUX_26_1</code>
<code>CORE_WINDOWS_27_1</code>
<code>CORE_LINUX_27_1</code>

Import detail

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

Attribute detail

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

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

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

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

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

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

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

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

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

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

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

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

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

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

```
GeometryContainers.CORE_WINDOWS_27_1 = (12, 'windows', 'core-windows-27.1')
```

```
GeometryContainers.CORE_LINUX_27_1 = (13, 'linux', 'core-linux-27.1')
```

Description

Module for connecting to a local Geometry Service Docker container.

Module detail

```
ansys.geometry.core.connection.docker_instance.get_geometry_container_type(instance: Local-  
DockerInstance)  
→ GeometryContainers | None
```

Provide back the GeometryContainers value.

Parameters

instance

[*LocalDockerInstance*] The LocalDockerInstance object.

Returns

GeometryContainers or *None*

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

Notes

This method returns the first hit on the available tags.

The launcher.py module

Summary

Functions

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

Description

Module for connecting to instances of the Geometry service.

Module detail

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

Start the Modeler interface for PyAnsys Geometry.

Parameters

mode

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

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

**kwargs

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

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

Returns

`ansys.geometry.core.modeler.Modeler`

Pythonic interface for geometry modeling.

Examples

Launch the Geometry service.

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

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

Start the Geometry service remotely using the PIM API.

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

Parameters

platform

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

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

version

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

client_log_level

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

client_log_file

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

**kwargs

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

Returns

`ansys.geometry.core.modeler.Modeler`

Instance of the Geometry service.

```
ansys.geometry.core.connection.launcher.launch_docker_modeler(port: int = py-
    geom_defaults.DEFAULT_PORT,
    connect_to_existing_service: bool
    = True, restart_if_existing_service:
    bool = False,
    name: str | None = None, image: an-
    sys.geometry.core.connection.docker_instance.Geome
    | None = None, client_log_level: int
    = logging.INFO, client_log_file:
    str | None = None, transport_mode:
    str | None = None, certs_dir:
    pathlib.Path | str | None = None,
    **kwargs: dict | None) → an-
    sys.geometry.core.modeler.Modeler
```

Start the Geometry service locally using Docker.

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

Parameters

port

[`int`, optional] Localhost port to deploy the Geometry service on or the the `Modeler`

interface to connect to (if it is already deployed). By default, the value is the one for the `DEFAULT_PORT` connection parameter.

connect_to_existing_service

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

restart_if_existing_service

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

name

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

image

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

client_log_level

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

client_log_file

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

transport_mode

[`str` | `None`] Transport mode selected, by default `None` and thus it will be selected for you based on the connection criteria. Options are: “insecure”, “mtls”.

certs_dir

[`Path` | `str` | `None`] Directory to use for TLS certificates. By default `None` and thus search for the “`ANSYS_GRPC_CERTIFICATES`” environment variable. If not found, it will use the “certs” folder assuming it is in the current working directory.

****kwargs**

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

Returns**Modeler**

Instance of the Geometry service.

```

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

```

Start Ansys Discovery remotely using the PIM API.

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

Parameters

version

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

client_log_level

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

client_log_file

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

**kwargs

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

Returns

`ansys.geometry.core.modeler.Modeler`

Instance of `Modeler`.

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

Start the Geometry service remotely using the PIM API.

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

Parameters

version

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

client_log_level

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

client_log_file

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

****kwargs**

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

Returns

`ansys.geometry.core.modeler.Modeler`

Instance of `Modeler`.

```

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

```

Start Ansys SpaceClaim remotely using the PIM API.

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

Parameters

version

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

client_log_level

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

client_log_file

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

**kwargs

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

Returns

`ansys.geometry.core.modeler.Modeler`

Instance of `Modeler`.

```
ansys.geometry.core.connection.launcher.launch_modeler_with_geometry_service(version: str | int
| None = None,
host: str =
'localhost',
port: int =
None,
enable_trace:
bool = False,
timeout: int =
60,
server_log_level:
int = 2,
client_log_level:
int =
logging.INFO,
server_logs_folder:
str = None,
client_log_file:
str = None,
server_working_dir:
str |
pathlib.Path |
None = None,
trans-
port_mode: str
| None = None,
uds_dir:
pathlib.Path |
str | None =
None, uds_id:
str | None =
None, certs_dir:
pathlib.Path |
str | None =
None,
proto_version:
str = None,
**kwargs: dict |
None) → an-
sys.geometry.core.modeler.Modeler
```

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

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

Parameters

version: str | int, optional

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

Ansys products versions and their corresponding int values:

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

host: str, optional

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

port

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

enable_trace

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

timeout

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

server_log_level

[int, optional]

Backend's log level from 0 to 3:

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

The default is 2 (Warning).

client_log_level

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

server_logs_folder

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

client_log_file

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

server_working_dir

[str | Path, optional] Sets the working directory for the product instance. If nothing is defined, the working directory will be inherited from the parent process.

transport_mode

[str | None] Transport mode selected, by default *None* and thus it will be selected for you based on the connection criteria. Options are: "insecure", "uds", "wnua", "mtls"

uds_dir

[Path | str | None] Directory to use for Unix Domain Sockets (UDS) transport mode. By default *None* and thus it will use the "~/conn" folder.

uds_id

[str | None] Optional ID to use for the UDS socket filename. By default *None* and thus it will use "aposdas_socket.sock". Otherwise, the socket filename will be "aposdas_socket-<uds_id>.sock".

certs_dir

[Path | str | None] Directory to use for TLS certificates. By default *None* and thus search for the "ANSYS_GRP_CERTIFICATES" environment variable. If not found, it will use the "certs" folder assuming it is in the current working directory.

proto_version

[str, default: None] The version of the gRPC API protocol to use. If None, the latest version supported by the server will be used. Options are "v0" and "v1".

****kwargs**

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

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

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

SystemError

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

Examples

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

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

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

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

```

ansys.geometry.core.connection.launcher.launch_modeler_with_discovery(version: str | int | None =
                                                                    None, host: str =
                                                                    'localhost', port: int =
                                                                    None, api_version: an-
                                                                    sys.geometry.core.connection.backend.ApiV
                                                                    = ApiVersions.LATEST,
                                                                    timeout: int = 150,
                                                                    manifest_path: str =
                                                                    None, hidden: bool =
                                                                    False, server_log_level:
                                                                    int = 2, client_log_level:
                                                                    int = logging.INFO,
                                                                    client_log_file: str =
                                                                    None,
                                                                    server_working_dir: str |
                                                                    pathlib.Path | None =
                                                                    None, transport_mode:
                                                                    str | None = None,
                                                                    uds_dir: pathlib.Path |
                                                                    str | None = None,
                                                                    uds_id: str | None =
                                                                    None, certs_dir:
                                                                    pathlib.Path | str | None
                                                                    = None, proto_version:
                                                                    str = None, **kwargs:
                                                                    dict | None)

```

Start Ansys Discovery locally using the `ProductInstance` class.

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

Parameters

version: str | int, optional

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

Ansys products versions and their corresponding int values:

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

host: str, optional

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

port

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

api_version: ApiVersions, optional

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

timeout

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

manifest_path

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

hidden

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

server_log_level

[[int](#), optional]

Backend's log level from 0 to 3:

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

The default is 2 (Warning).

client_log_level

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

client_log_file

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

server_working_dir

[[str](#) | Path, optional] Sets the working directory for the product instance. If nothing is defined, the working directory will be inherited from the parent process.

transport_mode

[[str](#) | [None](#)] Transport mode selected, by default *None* and thus it will be selected for you based on the connection criteria. Options are: "insecure", "uds", "wnua", "mtls"

uds_dir

[Path | [str](#) | [None](#)] Directory to use for Unix Domain Sockets (UDS) transport mode. By default *None* and thus it will use the "~/conn" folder.

uds_id

[[str](#) | [None](#)] Optional ID to use for the UDS socket filename. By default *None* and thus it will use "aposdas_socket.sock". Otherwise, the socket filename will be "aposdas_socket-<uds_id>.sock".

certs_dir

[Path | [str](#) | [None](#)] Directory to use for TLS certificates. By default *None* and thus search for the "ANSYS_GRPC_CERTIFICATES" environment variable. If not found, it will use the "certs" folder assuming it is in the current working directory.

proto_version

[[str](#), default: [None](#)] The version of the gRPC API protocol to use. If *None*, the latest version supported by the server will be used. Options are "v0" and "v1".

****kwargs**

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

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

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

SystemError:

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

Examples

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

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

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

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

```
ansys.geometry.core.connection.launcher.launch_modeler_with_spaceclaim(version: str | int | None
      = None, host: str =
      'localhost', port: int =
      None, api_version: an-
      sys.geometry.core.connection.backend.Api
      =
      ApiVersions.LATEST,
      timeout: int = 150,
      manifest_path: str =
      None, hidden: bool =
      False, server_log_level:
      int = 2,
      client_log_level: int =
      logging.INFO,
      client_log_file: str =
      None,
      server_working_dir: str
      | pathlib.Path | None =
      None, transport_mode:
      str | None = None,
      uds_dir: pathlib.Path |
      str | None = None,
      uds_id: str | None =
      None, certs_dir:
      pathlib.Path | str | None
      = None, proto_version:
      str = None, **kwargs:
      dict | None)
```

Start Ansys SpaceClaim locally using the ProductInstance class.

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

Parameters

version: str | int, optional

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

Ansys products versions and their corresponding int values:

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

host: str, optional

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

port

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

api_version: ApiVersions, optional

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

timeout

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

manifest_path

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

hidden

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

server_log_level

[`int`, optional]

Backend's log level from 0 to 3:

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

The default is 2 (Warning).

client_log_level

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

client_log_file

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

server_working_dir

[`str` | `Path`, optional] Sets the working directory for the product instance. If nothing is defined, the working directory will be inherited from the parent process.

transport_mode

[`str` | *None*] Transport mode selected, by default *None* and thus it will be selected for you based on the connection criteria. Options are: "insecure", "uds", "wnua", "mtls"

uds_dir

[`Path` | `str` | *None*] Directory to use for Unix Domain Sockets (UDS) transport mode. By default *None* and thus it will use the "`~/conn`" folder.

uds_id

[`str` | `None`] Optional ID to use for the UDS socket filename. By default *None* and thus it will use “aposdas_socket.sock”. Otherwise, the socket filename will be “aposdas_socket-<uds_id>.sock”.

certs_dir

[Path | `str` | `None`] Directory to use for TLS certificates. By default *None* and thus search for the “ANSYS_GRPC_CERTIFICATES” environment variable. If not found, it will use the “certs” folder assuming it is in the current working directory.

proto_version

[`str`, default: `None`] The version of the gRPC API protocol to use. If *None*, the latest version supported by the server will be used. Options are “v0” and “v1”.

****kwargs**

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

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

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

SystemError

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

Examples

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

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

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

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

```
ansys.geometry.core.connection.launcher.launch_modeler_with_core_service(version: str | int |  
                                                                           None = None, host:  
                                                                           str = 'localhost',  
                                                                           port: int = None,  
                                                                           enable_trace: bool =  
                                                                           False, timeout: int =  
                                                                           60, server_log_level:  
                                                                           int = 2,  
                                                                           client_log_level: int  
                                                                           = logging.INFO,  
                                                                           server_logs_folder:  
                                                                           str = None,  
                                                                           client_log_file: str =  
                                                                           None,  
                                                                           server_working_dir:  
                                                                           str | pathlib.Path |  
                                                                           None = None,  
                                                                           transport_mode: str |  
                                                                           None = None,  
                                                                           uds_dir: pathlib.Path  
                                                                           | str | None = None,  
                                                                           uds_id: str | None =  
                                                                           None, certs_dir:  
                                                                           pathlib.Path | str |  
                                                                           None = None,  
                                                                           proto_version: str =  
                                                                           None, **kwargs: dict  
                                                                           | None) → an-  
                                                                           sys.geometry.core.modeler.Modeler
```

Start the Geometry Core service locally using the ProductInstance class.

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

Parameters

version: str | int, optional

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

Ansys products versions and their corresponding int values:

- 252 : Ansys 25R2
- 261 : Ansys 26R1
- 271 : Ansys 27R1

host: str, optional

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

port

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

enable_trace

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

timeout

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

server_log_level

[[int](#), optional]

Backend's log level from 0 to 3:

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

The default is 2 (Warning).

client_log_level

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

server_logs_folder

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

client_log_file

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

server_working_dir

[[str](#) | Path, optional] Sets the working directory for the product instance. If nothing is defined, the working directory will be inherited from the parent process.

transport_mode

[[str](#) | [None](#)] Transport mode selected, by default *None* and thus it will be selected for you based on the connection criteria. Options are: "insecure", "uds", "wnua", "mtls"

uds_dir

[Path | [str](#) | [None](#)] Directory to use for Unix Domain Sockets (UDS) transport mode. By default *None* and thus it will use the "~/conn" folder.

uds_id

[[str](#) | [None](#)] Optional ID to use for the UDS socket filename. By default *None* and thus it will use "aposdas_socket.sock". Otherwise, the socket filename will be "aposdas_socket-<uds_id>.sock".

certs_dir

[Path | [str](#) | [None](#)] Directory to use for TLS certificates. By default *None* and thus search for the "ANSYS_GRPC_CERTIFICATES" environment variable. If not found, it will use the "certs" folder assuming it is in the current working directory.

proto_version

[[str](#), default: [None](#)] The version of the gRPC API protocol to use. If *None*, the latest version supported by the server will be used. Options are "v0" and "v1".

****kwargs**

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

Returns**Modeler**

Instance of the Geometry Core service.

Raises**ConnectionError**

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

SystemError

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

Examples

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

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

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

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

The product_instance.py module

Summary

Classes

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

Functions

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

Constants

<code>WINDOWS_GEOMETRY_SERVICE_FOLDER</code>	Default Geometry Service's folder name into the unified installer (DMS).
<code>CORE_GEOMETRY_SERVICE_FOLDER</code>	Default Geometry Service's folder name into the unified installer (Core Service).
<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 ADDINS_SUBFOLDER folder.
<code>MANIFEST_FILENAME</code>	Default backend's add-in filename.
<code>GEOMETRY_SERVICE_EXE</code>	The Windows Geometry Service's filename (DMS).
<code>CORE_GEOMETRY_SERVICE_EXE</code>	The Windows Geometry Service's filename (Core Service).
<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 add-in manifest file's path.
<code>BACKEND_SPACECLAIM_HIDDEN</code>	The argument to hide SpaceClaim's UI on the backend.
<code>BACKEND_SPACECLAIM_HIDDEN_ENVV</code>	SpaceClaim hidden backend's environment variable key.
<code>BACKEND_SPACECLAIM_HIDDEN_ENVV</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 add-in manifest file's path.
<code>ANSYS_GEOMETRY_SERVICE_ROOT</code>	Local Geometry Service install location. This is for GeometryService and CoreGeometryService.

ProductInstance

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

ProductInstance class.

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

Parameters

pid

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

Overview

Methods

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

Import detail

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

Method detail

`ProductInstance.close() → bool`

Close the process associated to the pid.

Description

Module containing the `ProductInstance` class.

Module detail

```

ansys.geometry.core.connection.product_instance.prepare_and_start_backend(backend_type: an-
sys.geometry.core.connection.backend
version: str | int =
None, host: str =
'localhost', port: int
= None,
enable_trace: bool
= False,
api_version: an-
sys.geometry.core.connection.backend
= ApiVer-
sions.LATEST,
timeout: int = 150,
manifest_path: str
= None, hidden:
bool = False,
server_log_level:
int = 2,
client_log_level: int
= logging.INFO,
server_logs_folder:
str = None,
client_log_file: str
= None,
transport_mode: str
| None = None,
uds_dir:
pathlib.Path | str |
None = None,
uds_id: str | None =
None, certs_dir:
pathlib.Path | str |
None = None, spe-
cific_minimum_version:
int = None,
server_working_dir:
str | pathlib.Path |
None = None,
proto_version: str
= None) → an-
sys.geometry.core.modeler.Modeler

```

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

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

Parameters

version

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

host: str, optional

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

localhost.

port

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

enable_trace

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

api_version: ``ApiVersions``, optional

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

timeout

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

manifest_path

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

hidden

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

server_log_level

[[int](#), optional]

Backend's log level from 0 to 3:

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

The default is 2 (Warning).

client_log_level

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

server_logs_folder

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

client_log_file

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

transport_mode

[[str](#) | `None`] Transport mode selected, by default *None* and thus it will be selected for you based on the connection criteria. Options are: "insecure", "uds", "wnua", "mtls"

uds_dir

[Path | [str](#) | `None`] Directory to use for Unix Domain Sockets (UDS) transport mode. By default *None* and thus it will use the "~/conn" folder.

uds_id

[[str](#) | `None`] Optional ID to use for the UDS socket filename. By default *None* and thus it will use "apodas_socket.sock". Otherwise, the socket filename will be "apodas_socket-<uds_id>.sock".

certs_dir

[Path | [str](#) | `None`] Directory to use for TLS certificates. By default *None* and thus search for the "ANSYS_GRP_CERTIFICATES" environment variable. If not found, it will use the "certs" folder assuming it is in the current working directory.

specific_minimum_version

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

server_working_dir

[**str** | Path, optional] Sets the working directory for the product instance. If nothing is defined, the working directory will be inherited from the parent process.

proto_version: str, optional

The version of the gRPC API protocol to use. If None, the latest version supported by the server will be used. Options are “v0” and “v1”.

Returns**Modeler**

Instance of the Geometry service.

Raises**ConnectionError**

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

SystemError

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

`ansys.geometry.core.connection.product_instance.get_available_port() → int`

Return an available port to be used.

Returns**int**

The available port.

`ansys.geometry.core.connection.product_instance.WINDOWS_GEOMETRY_SERVICE_FOLDER = 'GeometryService'`

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

`ansys.geometry.core.connection.product_instance.CORE_GEOMETRY_SERVICE_FOLDER = 'GeometryService'`

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

`ansys.geometry.core.connection.product_instance.DISCOVERY_FOLDER = 'Discovery'`

Default Discovery’s folder name into the unified installer.

`ansys.geometry.core.connection.product_instance.SPACECLAIM_FOLDER = 'scdm'`

Default SpaceClaim’s folder name into the unified installer.

`ansys.geometry.core.connection.product_instance.ADDINS_SUBFOLDER = 'Addins'`

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

`ansys.geometry.core.connection.product_instance.BACKEND_SUBFOLDER = 'ApiServer'`

Default backend’s folder name into the ADDINS_SUBFOLDER folder.

`ansys.geometry.core.connection.product_instance.MANIFEST_FILENAME = 'Presentation.ApiServerAddIn.Manifest.xml'`

Default backend’s add-in filename.

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

```
ansys.geometry.core.connection.product_instance.GEOMETRY_SERVICE_EXE =
'Presentation.ApiServerDMS.exe'
```

The Windows Geometry Service's filename (DMS).

```
ansys.geometry.core.connection.product_instance.CORE_GEOMETRY_SERVICE_EXE =
'Presentation.ApiServerCoreService.exe'
```

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

```
ansys.geometry.core.connection.product_instance.DISCOVERY_EXE = 'Discovery.exe'
```

The Ansys Discovery's filename.

```
ansys.geometry.core.connection.product_instance.SPACECLAIM_EXE = 'SpaceClaim.exe'
```

The Ansys SpaceClaim's filename.

```
ansys.geometry.core.connection.product_instance.BACKEND_LOG_LEVEL_VARIABLE = 'LOG_LEVEL'
```

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

```
ansys.geometry.core.connection.product_instance.BACKEND_TRACE_VARIABLE = 'ENABLE_TRACE'
```

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

```
ansys.geometry.core.connection.product_instance.BACKEND_HOST_VARIABLE = 'API_ADDRESS'
```

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

```
ansys.geometry.core.connection.product_instance.BACKEND_PORT_VARIABLE = 'API_PORT'
```

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

```
ansys.geometry.core.connection.product_instance.BACKEND_LOGS_FOLDER_VARIABLE =
'ANS_DSCO_REMOTE_LOGS_FOLDER'
```

The backend's logs folder path to be used.

Only applicable to the Ansys Geometry Service.

```
ansys.geometry.core.connection.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.

```
ansys.geometry.core.connection.product_instance.BACKEND_SPACECLAIM_OPTIONS =
'--spaceclaim-options'
```

The additional argument for local Ansys Discovery start.

To be used only with Ansys Discovery.

```
ansys.geometry.core.connection.product_instance.BACKEND_ADDIN_MANIFEST_ARGUMENT =
'/ADDINMANIFESTFILE='
```

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

To be used only with Ansys Discovery and Ansys SpaceClaim.

```
ansys.geometry.core.connection.product_instance.BACKEND_SPACECLAIM_HIDDEN =
'/Headless=True'
```

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

To be used only with Ansys SpaceClaim.

```
ansys.geometry.core.connection.product_instance.BACKEND_SPACECLAIM_HIDDEN_ENVVAR_KEY =
'SPACECLAIM_MODE'
```

SpaceClaim hidden backend's environment variable key.

To be used only with Ansys SpaceClaim.

```
ansys.geometry.core.connection.product_instance.BACKEND_SPACECLAIM_HIDDEN_ENVVAR_VALUE =
'2'
```

SpaceClaim hidden backend's environment variable value.

To be used only with Ansys SpaceClaim.

```
ansys.geometry.core.connection.product_instance.BACKEND_DISCOVERY_HIDDEN = '--hidden'
```

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

To be used only with Ansys Discovery.

```
ansys.geometry.core.connection.product_instance.BACKEND_SPLASH_OFF = '/Splash=False'
```

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

To be used only with Ansys Discovery and Ansys SpaceClaim.

```
ansys.geometry.core.connection.product_instance.ANSYS_GEOMETRY_SERVICE_ROOT =
'ANSYS_GEOMETRY_SERVICE_ROOT'
```

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

The validate.py module

Summary

Functions

<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

```
ansys.geometry.core.connection.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>datumplane</i>	Module for creating and managing datum planes.
<i>design</i>	Provides for managing designs.
<i>designpoint</i>	Module for creating and managing design points.
<i>edge</i>	Module for managing an edge.
<i>face</i>	Module for managing a face.
<i>geometry_commands</i>	Provides tools for pulling geometry.
<i>mating_conditions</i>	Provides dataclasses for mating conditions.
<i>part</i>	Module providing fundamental data of an assembly.
<i>selection</i>	Module for creating a named selection.
<i>vertex</i>	Module for managing a vertex.

The beam.py module

Summary

Classes

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

Enums

<i>BeamType</i>	Provides values for the beam types supported.
<i>SectionAnchorType</i>	Provides values for the section anchor types supported.

BeamProfile

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

Represents a single beam profile organized within the design assembly.

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

Parameters

id

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

name

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

Notes

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

Overview

Properties

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

Import detail

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

Property detail

property BeamProfile.id: **str**

ID of the beam profile.

property BeamProfile.name: **str**

Name of the beam profile.

BeamCircularProfile

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

Bases: BeamProfile

Represents a single circular beam profile.

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

Parameters

id

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

name

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

radius

[Distance] Radius of the circle.

center: Point3D

3D point representing the center of the circle.

direction_x: UnitVector3D

X-axis direction.

direction_y: UnitVector3D
Y-axis direction.

Notes

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

Overview

Properties

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

Special methods

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

Import detail

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

Property detail

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

Radius of the circular beam profile.

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

Center of the circular beam profile.

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

X-axis direction of the circular beam profile.

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

Y-axis direction of the circular beam profile.

Method detail

BeamCircularProfile.__repr__() → str

Represent the BeamCircularProfile as a string.

BeamCrossSectionInfo

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

Represents the cross-section information for a beam.

Parameters

section_anchor

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

section_angle

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

section_frame

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

section_profile

[BeamProfile] The section profile in the XY plane.

Overview

Properties

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

Special methods

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

Import detail

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

Property detail

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

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

property BeamCrossSectionInfo.**section_angle**: `float`

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

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

The section frame at the start of the beam.

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

The section profile in the XY plane.

Method detail

BeamCrossSectionInfo.__repr__() → str

Represent the BeamCrossSectionInfo as a string.

BeamProperties

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

Represents the properties of a beam.

Parameters

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

Overview

Properties

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

Import detail

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

Property detail

property BeamProperties.area: float

The cross-sectional area of the beam.

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

The centroid of the beam section.

property BeamProperties.warping_constant: float

The warping constant of the beam.

property BeamProperties.ixx: float

The moment of inertia about the x-axis.

property BeamProperties.iyx: float

The product of inertia.

property BeamProperties.iyy: float

The moment of inertia about the y-axis.

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

The shear center of the beam.

property BeamProperties.torsion_constant: float

The torsion constant of the beam.

Beam

```
class ansys.geometry.core.designer.beam.Beam(id: str, start: ansys.geometry.core.math.point.Point3D,
                                             end: ansys.geometry.core.math.point.Point3D, profile:
                                             BeamProfile | None, parent_component:
                                             ansys.geometry.core.designer.component.Component,
                                             name: str = None, is_deleted: bool = False, is_reversed:
                                             bool = False, is_rigid: bool = False, material:
                                             ansys.geometry.core.materials.material.Material = None,
                                             cross_section: BeamCrossSectionInfo = None,
                                             properties: BeamProperties = None, shape: an-
                                             sys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve
                                             = None, beam_type: BeamType = None)
```

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

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

Parameters

id

[str] Server-defined ID for the body.

name

[str] User-defined label for the body.

start

[Point3D] Start of the beam line segment.

end

[Point3D] End of the beam line segment.

profile

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

parent_component

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

Overview

Methods

<code>get_named_selections</code>	Get the named selections that include this beam.
-----------------------------------	--

Properties

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

Special methods

<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: ansys.geometry.core.designer.component.Component`

Component node that the beam is under.

property `Beam.is_alive: bool`

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

Method detail

`Beam.get_named_selections()` → `list[ansys.geometry.core.designer.selection.NamedSelection]`

Get the named selections that include this beam.

Returns

`list[NamedSelection]`

List of named selections that include this beam.

`Beam.__repr__()` → `str`

Represent the beam as a string.

BeamType

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

Bases: `enum.Enum`

Provides values for the beam types supported.

Overview

Attributes

<code>BEAM</code>
<code>SPRING</code>
<code>LINK_TRUSS</code>
<code>CABLE</code>
<code>PIPE</code>
<code>THERMALFLUID</code>
<code>UNKNOWN</code>

Import detail

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

Attribute detail

`BeamType.BEAM = 0`

`BeamType.SPRING = 1`

`BeamType.LINK_TRUSS = 2`

`BeamType.CABLE = 3`

`BeamType.PIPE = 4`

`BeamType.THERMALFLUID = 5`

`BeamType.UNKNOWN = 6`

SectionAnchorType

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

Bases: `enum.Enum`

Provides values for the section anchor types supported.

Overview

Attributes

<i>CENTROID</i>
<i>SHEAR_CENTER</i>
<i>ANCHOR_LOCATION</i>

Import detail

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

Attribute detail

`SectionAnchorType.CENTROID = 0`

`SectionAnchorType.SHEAR_CENTER = 1`

`SectionAnchorType.ANCHOR_LOCATION = 2`

Description

Provides for creating and managing a beam.

The body.py module

Summary

Interfaces

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

Classes

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

Enums

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

Constants

`__TEMPORARY_BOOL_OPS_FIX__`

IBody

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

Bases: `abc.ABC`

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

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

Overview

Abstract methods

<code>id</code>	Get the ID of the body as a string.
<code>name</code>	Get the name of the body.
<code>set_name</code>	Set the name of the body.
<code>fill_style</code>	Get the fill style of the body.
<code>set_fill_style</code>	Set the fill style of the body.
<code>is_suppressed</code>	Get the body suppression state.
<code>set_suppressed</code>	Set the body suppression state.
<code>color</code>	Get the color of the body.
<code>opacity</code>	Get the float value of the opacity for the body.
<code>set_color</code>	Set the color of the body.
<code>faces</code>	Get a list of all faces within the body.
<code>edges</code>	Get a list of all edges within the body.
<code>vertices</code>	Get a list of all vertices within the body.
<code>is_alive</code>	Check if the body is still alive and has not been deleted.
<code>is_surface</code>	Check if the body is a planar body.
<code>surface_thickness</code>	Get the surface thickness of a surface body.
<code>surface_offset</code>	Get the surface offset type of a surface body.
<code>volume</code>	Calculate the volume of the body.
<code>material</code>	Get the assigned material of the body.
<code>bounding_box</code>	Get the bounding box of the body.
<code>get_bounding_box</code>	Get the bounding box of the body.
<code>assign_material</code>	Assign a material against the active design.
<code>get_assigned_material</code>	Get the assigned material of the body.
<code>remove_assigned_material</code>	Remove the material assigned to the body.
<code>add_midsurface_thickness</code>	Add a mid-surface thickness to a surface body.
<code>add_midsurface_offset</code>	Add a mid-surface offset to a surface body.
<code>imprint_curves</code>	Imprint all specified geometries onto specified faces of the body.
<code>project_curves</code>	Project all specified geometries onto the body.
<code>imprint_projected_curves</code>	Project and imprint specified geometries onto the body.
<code>translate</code>	Translate the body in a specified direction and distance.
<code>rotate</code>	Rotate the geometry body around the specified axis by a given angle.
<code>scale</code>	Scale the geometry body by the given value.

continues on next page

Table 1 – continued from previous page

<i>map</i>	Map the geometry body to the new specified frame.
<i>mirror</i>	Mirror the geometry body across the specified plane.
<i>get_collision</i>	Get the collision state between bodies.
<i>copy</i>	Create a copy of the body under the specified parent.
<i>get_named_selections</i>	Get the named selections associated with the body.
<i>get_raw_tessellation</i>	Tessellate the body and return the raw tessellation data.
<i>tessellate</i>	Tessellate the body and return the geometry as triangles.
<i>get_vtk_tessellation</i>	Build VTK objects from raw tessellation data.
<i>shell_body</i>	Shell the body to the thickness specified.
<i>remove_faces</i>	Shell by removing a given set of faces.
<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.
<i>combine_merge</i>	Combine this body with another body or bodies, merging them into a single body.

Import detail

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

Method detail

abstractmethod `IBody.id() → str`

Get the ID of the body as a string.

abstractmethod `IBody.name() → str`

Get the name of the body.

abstractmethod `IBody.set_name(str) → None`

Set the name of the body.

Warning

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

abstractmethod `IBody.fill_style() → FillStyle`

Get the fill style of the body.

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

Set the fill style of the body.

Warning

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

abstractmethod `IBody.is_suppressed() → bool`

Get the body suppression state.

Warning

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

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

Set the body suppression state.

Warning

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

abstractmethod `IBody.color() → str`

Get the color of the body.

abstractmethod `IBody.opacity() → float`

Get the float value of the opacity for the body.

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

Set the color of the body.

Parameters

color

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

Warning

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

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

Get a list of all faces within the body.

Returns

`list[Face]`

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

Get a list of all edges within the body.

Returns

`list[Edge]`

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

Get a list of all vertices within the body.

Returns

`list[Vertex]`

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

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

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

Check if the body is a planar body.

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

Get the surface thickness of a surface body.

Notes

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

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

Get the surface offset type of a surface body.

Notes

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

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

Calculate the volume of the body.

Notes

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

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

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

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

Get the bounding box of the body.

Returns

BoundingBox

Bounding box of the body.

Warning

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

abstractmethod `IBody.get_bounding_box(tight: bool = False)` → `ansys.geometry.core.math.bbox.BoundingBox`

Get the bounding box of the body.

Parameters

tight

[`bool`, default: `False`] Whether to use a tight tolerance when calculating the bounding box.

Returns

BoundingBox

Bounding box of the body.

Warning

This method is only available starting on Ansys release 27R1.

abstractmethod `IBody.assign_material(material: ansys.geometry.core.materials.material.Material) → None`
Assign a material against the active design.

Parameters**material**

[Material] Source material data.

abstractmethod `IBody.get_assigned_material() → ansys.geometry.core.materials.material.Material`
Get the assigned material of the body.

Returns**Material**

Material assigned to the body.

abstractmethod `IBody.remove_assigned_material() → None`
Remove the material assigned to the body.

abstractmethod `IBody.add_midsurface_thickness(thickness: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real) → None`

Add a mid-surface thickness to a surface body.

Parameters**thickness**

[Distance | Quantity | Real] Thickness to assign.

Notes

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

abstractmethod `IBody.add_midsurface_offset(offset: MidSurfaceOffsetType) → None`
Add a mid-surface offset to a surface body.

Parameters**offset_type**

[MidSurfaceOffsetType] Surface offset to assign.

Notes

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

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

Imprint all specified geometries onto specified faces of the body.

Parameters**faces:** `list[Face]`

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

sketch: `Sketch`

All curves to imprint on the faces.

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

All impacted edges and faces from the imprint operation.

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

Project all specified geometries onto the body.

Parameters**direction:** `UnitVector3D`

Direction of the projection.

sketch: `Sketch`

All curves to project on the body.

closest_face: `bool`

Whether to target the closest face with the projection.

only_one_curve: `bool`, **default:** `False`Whether to project only one curve of the entire sketch. When `True`, only one curve is projected.**Returns**`list[Face]`

All faces from the project curves operation.

Notes

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

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

Project and imprint specified geometries onto the body.

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

Parameters**direction:** `UnitVector3D`

Direction of the projection.

sketch: `Sketch`

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

`list[Face]`

All imprinted faces from the operation.

Notes

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

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

Translate the body in a specified direction and distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Distance (magnitude) of the translation.

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

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

Parameters

axis_origin: Point3D

Origin of the rotational axis.

axis_direction: UnitVector3D

The axis of rotation.

angle: ~pint.Quantity | Angle | Real

Angle (magnitude) of the rotation.

Warning

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

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

Scale the geometry body by the given value.

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

Parameters

value: Real

Value to scale the body by.

Warning

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

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

Map the geometry body to the new specified frame.

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

Parameters

frame: Frame

Structure defining the orientation of the body.

Warning

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

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

Mirror the geometry body across the specified plane.

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

Parameters

plane: Plane

Represents the mirror.

Warning

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

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

Get the collision state between bodies.

Parameters

body: Body

Object that the collision state is checked with.

Returns

CollisionType

Enum that defines the collision state between bodies.

Warning

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

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

Create a copy of the body under the specified parent.

Parameters

parent: Component

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

name: str

Name to give the new body.

Returns

Body

Copy of the body.

abstractmethod `IBody.get_named_selections()` → `list[ansys.geometry.core.designer.selection.NamedSelection]`

Get the named selections associated with the body.

Returns

`list[NamedSelection]`

List of named selections associated with the body.

abstractmethod `IBody.get_raw_tessellation(transform: ansys.geometry.core.math.matrix.Matrix44 = IDENTITY_MATRIX44, tess_options: ansys.geometry.core.misc.options.TessellationOptions | None = None, reset_cache: bool = False, include_faces: bool = True, include_edges: bool = False) → dict`

Tessellate the body and return the raw tessellation data.

Parameters

transform

[`Matrix44`, default: `IDENTITY_MATRIX44`] A transformation matrix to apply to the tessellation.

tess_options

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

reset_cache

[`bool`, default: `False`] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[`bool`, default: `True`] Whether to include face tessellation data in the output.

include_edges

[`bool`, default: `False`] Whether to include edge tessellation data in the output.

Returns

`dict`

Dictionary with face and edge IDs as keys and face and edge tessellation data as values.

abstractmethod `IBody.tessellate(merge: bool = False, tess_options: ansys.geometry.core.misc.options.TessellationOptions | None = None, reset_cache: bool = False, include_faces: bool = True, include_edges: bool = False) → pyvista.PolyData | pyvista.MultiBlock`

Tessellate the body and return the geometry as triangles.

Parameters

merge

[`bool`, default: `False`] Whether to merge the body into a single mesh. When `False` (default),

the number of triangles are preserved and only the topology is merged. When True, the individual faces of the tessellation are merged.

tess_options

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

reset_cache

[bool, default: False] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[bool, default: True] Whether to include face tessellation data in the output.

include_edges

[bool, default: False] Whether to include edge tessellation data in the output.

Returns

PolyData, MultiBlock

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

Examples

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

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

Merge the body:

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

abstractmethod `IBody.get_vtk_tessellation`(*merge*: *bool* = *False*, *tess_options*: *ansys.geometry.core.misc.options.TessellationOptions* | *None* = *None*, *reset_cache*: *bool* = *False*, *include_faces*: *bool* = *True*, *include_edges*: *bool* = *False*, *_raw_tessellation*: *dict* | *None* = *None*) → *vtkmodules.vtkCommonDataModel.vtkPolyData* | *vtkmodules.vtkCommonDataModel.vtkMultiBlockDataSet*

Build VTK objects from raw tessellation data.

Parameters

merge

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

tess_options

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

reset_cache

[*bool*, default: *False*] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[*bool*, default: *True*] Whether to include face tessellation data in the output.

include_edges

[*bool*, default: *False*] Whether to include edge tessellation data in the output.

_raw_tessellation

[*dict* | *None*, default: *None*] Raw tessellation data with face and edge IDs as keys and tessellation data as values.. If *None*, the method requests the raw tessellation data from the server. This parameter is intended for internal use only.

Returns

vtkPolyData, vtkMultiBlockDataSet

Merged *vtkPolyData* if *merge=True* or a *vtkMultiBlockDataSet*.

abstractmethod `IBody.shell_body`(*offset*: *ansys.geometry.core.misc.measurements.Distance* | *pint.Quantity* | *ansys.geometry.core.typing.Real*) → *bool*

Shell the body to the thickness specified.

Parameters

offset

[*Distance* | *Quantity* | *Real*] Shell thickness.

Returns

bool

True when successful, False when failed.

Warning

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

abstractmethod `IBody.remove_faces(selection: ansys.geometry.core.designer.face.Face | collections.abc.Iterable[ansys.geometry.core.designer.face.Face], offset: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real) → bool`

Shell by removing a given set of faces.

Parameters

selection

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

offset

[Distance | Quantity | Real] Shell thickness.

Returns

bool

True when successful, False when failed.

Warning

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

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

Plot the body.

Parameters

merge

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

screenshot

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

use_frame

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

use_service_colors

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

****plotting_options**

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

Examples

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

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
```

(continues on next page)

(continued from previous page)

```

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

```

Plot the body and color each face individually:

```

>>> body.plot(multi_colors=True)

```

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

Intersect two (or more) bodies.

Parameters

other

[[Body](#)] Body to intersect with.

keep_other

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

Raises

ValueError

If the bodies do not intersect.

Notes

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

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

Subtract two (or more) bodies.

Parameters

other

[[Body](#)] Body to subtract from the *self* parameter.

keep_other

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

Raises

ValueError

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

Notes

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

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

Unite two (or more) bodies.

Parameters

other

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

keep_other

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

Notes

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

`IBody.combine_merge(other: Body | list[Body]) → None`

Combine this body with another body or bodies, merging them into a single body.

Parameters

other

[Union[Body, list[Body]]] The body or list of bodies to combine with this body.

Notes

The `self` parameter is directly modified, and the `other` bodies are consumed.

MasterBody

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

Bases: `IBody`

Represents solids and surfaces organized within the design assembly.

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

Parameters

id

[str] Server-defined ID for the body.

name

[str] User-defined label for the body.

parent_component

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

grpc_client

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

is_surface

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

Overview

Abstract methods

<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>copy</i>	Create a copy of the body under the specified parent.
<i>get_named_selections</i>	Get the named selections associated with the body.
<i>plot</i>	Plot the body.
<i>intersect</i>	Intersect two (or more) bodies.
<i>subtract</i>	Subtract two (or more) bodies.
<i>unite</i>	Unite two (or more) bodies.

Methods

<i>reset_tessellation_cache</i>	Decorate MasterBody methods that need tessellation cache update.
<i>get_bounding_box</i>	Get the bounding box of the body.
<i>assign_material</i>	Assign a material against the active design.
<i>get_assigned_material</i>	Get the assigned material of the body.
<i>remove_assigned_material</i>	Remove the material assigned to the body.
<i>add_midsurface_thickness</i>	Add a mid-surface thickness to a surface body.
<i>add_midsurface_offset</i>	Add a mid-surface offset to a surface body.
<i>translate</i>	Translate the body in a specified direction and distance.
<i>set_name</i>	Set the name of the body.
<i>set_fill_style</i>	Set the fill style of the body.
<i>set_suppressed</i>	Set the body suppression state.
<i>set_color</i>	Set the color of the body.
<i>set_opacity</i>	Set the opacity of the body.
<i>rotate</i>	Rotate the geometry body around the specified axis by a given angle.
<i>scale</i>	Scale the geometry body by the given value.
<i>map</i>	Map the geometry body to the new specified frame.
<i>mirror</i>	Mirror the geometry body across the specified plane.
<i>get_collision</i>	Get the collision state between bodies.
<i>get_raw_tessellation</i>	Tessellate the body and return the raw tessellation data.
<i>tessellate</i>	Tessellate the body and return the geometry as triangles.
<i>get_vtk_tessellation</i>	Build VTK objects from raw tessellation data.
<i>shell_body</i>	Shell the body to the thickness specified.
<i>remove_faces</i>	Shell by removing a given set of faces.
<i>combine_merge</i>	Combine this body with another body or bodies, merging them into a single body.

Properties

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

Special methods

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

Import detail

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

Property detail

property `MasterBody.id: str`

Get the ID of the body as a string.

property `MasterBody.name: str`

Get the name of the body.

property `MasterBody.fill_style: str`

Get the fill style of the body.

property `MasterBody.is_suppressed: bool`

Get the body suppression state.

Warning

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

property `MasterBody.color: str`

Get the current color of the body.

property `MasterBody.opacity: float`

Get the float value of the opacity for the body.

property MasterBody.is_surface: bool

Check if the body is a planar body.

property MasterBody.surface_thickness: pint.Quantity | None

Get the surface thickness of a surface body.

Notes

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

property MasterBody.surface_offset: MidSurfaceOffsetType | None

Get the surface offset type of a surface body.

Notes

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

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

Get a list of all faces within the body.

Returns

list[Face]

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

Get a list of all edges within the body.

Returns

list[Edge]

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

Get a list of all vertices within the body.

Returns

list[Vertex]

property MasterBody.is_alive: bool

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

property MasterBody.volume: pint.Quantity

Calculate the volume of the body.

Notes

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

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

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

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

Get the bounding box of the body.

Returns

BoundingBox

Bounding box of the body.

Warning

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

Method detail

`MasterBody.reset_tessellation_cache()`

Decorate MasterBody methods that need tessellation cache update.

Parameters

func

[method] Method to call.

Returns

Any

Output of the method, if any.

`MasterBody.get_bounding_box(tight: bool = False) → ansys.geometry.core.math.bbox.BoundingBox`

Get the bounding box of the body.

Parameters

tight

[bool, default: False] Whether to use a tight tolerance when calculating the bounding box.

Returns

BoundingBox

Bounding box of the body.

Warning

This method is only available starting on Ansys release 27R1.

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

Assign a material against the active design.

Parameters

material

[Material] Source material data.

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

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

`MasterBody.remove_assigned_material() → None`

Remove the material assigned to the body.

MasterBody.add_midsurface_thickness(*thickness*: `ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real`) → `None`

Add a mid-surface thickness to a surface body.

Parameters

thickness

[`Distance | Quantity | Real`] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters

offset_type

[`MidSurfaceOffsetType`] Surface offset to assign.

Notes

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

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

Imprint all specified geometries onto specified faces of the body.

Parameters

faces: `list[Face]`

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

sketch: `Sketch`

All curves to imprint on the faces.

Returns

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

All impacted edges and faces from the imprint operation.

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

Project all specified geometries onto the body.

Parameters

direction: `UnitVector3D`

Direction of the projection.

sketch: `Sketch`

All curves to project on the body.

closest_face: `bool`

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

`list[Face]`

All faces from the project curves operation.

Notes

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

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

Project and imprint specified geometries onto the body.

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

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

`list[Face]`

All imprinted faces from the operation.

Notes

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

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

Translate the body in a specified direction and distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: `~pint.Quantity | Distance | Real`

Distance (magnitude) of the translation.

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

Set the name of the body.

Warning

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

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

Set the fill style of the body.

Warning

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

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

Set the body suppression state.

Warning

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

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

Set the color of the body.

Parameters

color

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

Warning

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

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

Set the opacity of the body.

Warning

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

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

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

Parameters**axis_origin: Point3D**

Origin of the rotational axis.

axis_direction: UnitVector3D

The axis of rotation.

angle: ~pint.Quantity | Angle | Real

Angle (magnitude) of the rotation.

Warning

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

MasterBody.scale(value: [ansys.geometry.core.typing.Real](#)) → None

Scale the geometry body by the given value.

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

Parameters**value: Real**

Value to scale the body by.

Warning

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

MasterBody.map(frame: [ansys.geometry.core.math.frame.Frame](#)) → None

Map the geometry body to the new specified frame.

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

Parameters**frame: Frame**

Structure defining the orientation of the body.

Warning

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

MasterBody.mirror(plane: [ansys.geometry.core.math.plane.Plane](#)) → None

Mirror the geometry body across the specified plane.

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

Parameters**plane: Plane**

Represents the mirror.

Warning

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

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

Get the collision state between bodies.

Parameters

body: Body

Object that the collision state is checked with.

Returns

CollisionType

Enum that defines the collision state between bodies.

Warning

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

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

Create a copy of the body under the specified parent.

Parameters

parent: Component

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

name: str

Name to give the new body.

Returns

Body

Copy of the body.

abstractmethod `MasterBody.get_named_selections() → list[ansys.geometry.core.designer.selection.NamedSelection]`

Get the named selections associated with the body.

Returns

list[NamedSelection]

List of named selections associated with the body.

`MasterBody.get_raw_tessellation(tess_options: ansys.geometry.core.misc.options.TessellationOptions | None = None, reset_cache: bool = False, include_faces: bool = True, include_edges: bool = False) → dict`

Tessellate the body and return the raw tessellation data.

Parameters

transform

[Matrix44, default: IDENTITY_MATRIX44] A transformation matrix to apply to the tessellation.

tess_options

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

reset_cache

[bool, default: **False**] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[bool, default: **True**] Whether to include face tessellation data in the output.

include_edges

[bool, default: **False**] Whether to include edge tessellation data in the output.

Returns**dict**

Dictionary with face and edge IDs as keys and face and edge tessellation data as values.

```
MasterBody.tessellate(merge: bool = False, transform: ansys.geometry.core.math.matrix.Matrix44 =
    IDENTITY_MATRIX44, tess_options:
    ansys.geometry.core.misc.options.TessellationOptions | None = None, reset_cache: bool
    = False, include_faces: bool = True, include_edges: bool = False) → pyvista.PolyData
    | pyvista.MultiBlock
```

Tessellate the body and return the geometry as triangles.

Parameters**merge**

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

tess_options

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

reset_cache

[bool, default: **False**] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[bool, default: **True**] Whether to include face tessellation data in the output.

include_edges

[bool, default: **False**] Whether to include edge tessellation data in the output.

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
```

(continues on next page)

(continued from previous page)

```

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

```

Merge the body:

```

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

```

`MasterBody.get_vtk_tessellation`(*merge: bool = False, tess_options:*
 ansys.geometry.core.misc.options.TessellationOptions | None = None,
 reset_cache: bool = False, include_faces: bool = True, include_edges:
 bool = False, _raw_tessellation: dict | None = None) →
 vtkmodules.vtkCommonDataModel.vtkPolyData |
 vtkmodules.vtkCommonDataModel.vtkMultiBlockDataSet

Build VTK objects from raw tessellation data.

Parameters

merge

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

tess_options

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

reset_cache

[*bool*, default: *False*] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[*bool*, default: *True*] Whether to include face tessellation data in the output.

include_edges

[*bool*, default: *False*] Whether to include edge tessellation data in the output.

_raw_tessellation

[**dict** | **None**, default: **None**] Raw tessellation data with face and edge IDs as keys and tessellation data as values.. If **None**, the method requests the raw tessellation data from the server. This parameter is intended for internal use only.

Returns

vtkPolyData, **vtkMultiBlockDataSet**

Merged *vtkPolyData* if `merge=True` or a *vtkMultiBlockDataSet*.

MasterBody.shell_body(*offset*: [ansys.geometry.core.misc.measurements.Distance](#) | [pint.Quantity](#) | [ansys.geometry.core.typing.Real](#)) → **bool**

Shell the body to the thickness specified.

Parameters

offset

[[Distance](#) | [Quantity](#) | [Real](#)] Shell thickness.

Returns

bool

True when successful, False when failed.

Warning

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

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

Shell by removing a given set of faces.

Parameters

selection

[[Face](#) | [Iterable](#)[[Face](#)]] Face or faces to be removed.

offset

[[Distance](#) | [Quantity](#) | [Real](#)] Shell thickness.

Returns

bool

True when successful, False when failed.

Warning

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

MasterBody.combine_merge(*other*: [Body](#) | [list](#)[[Body](#)]) → **None**

Combine this body with another body or bodies, merging them into a single body.

Parameters

other

[[Union](#)[[Body](#), [list](#)[[Body](#)]]] The body or list of bodies to combine with this body.

Notes

The self parameter is directly modified, and the other bodies are consumed.

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

Plot the body.

Parameters

merge

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

screenshot

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

use_frame

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

use_service_colors

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

**plotting_options

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

Examples

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

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

Plot the body and color each face individually:

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

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

Intersect two (or more) bodies.

Parameters

other

[Body] Body to intersect with.

keep_other

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

Raises**ValueError**

If the bodies do not intersect.

Notes

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

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

Subtract two (or more) bodies.

Parameters**other**

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

keep_other

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

Raises**ValueError**

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

Notes

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

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

Unite two (or more) bodies.

Parameters**other**

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

keep_other

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

Notes

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

`MasterBody.__repr__() → str`

Represent the master body as a string.

Body

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

Bases: `IBody`

Represents solids and surfaces organized within the design assembly.

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

Parameters

id

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

name

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

parent_component

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

template

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

Overview

Methods

<code>reset_tessellation_cache</code>	Decorate Body methods that require a tessellation cache update.
<code>get_bounding_box</code>	Get the bounding box of the body.
<code>assign_material</code>	Assign a material against the active design.
<code>get_assigned_material</code>	Get the assigned material of the body.
<code>remove_assigned_material</code>	Remove the material assigned to the body.
<code>add_midsurface_thickness</code>	Add a mid-surface thickness to a surface body.
<code>add_midsurface_offset</code>	Add a mid-surface offset to a surface body.
<code>imprint_curves</code>	Imprint all specified geometries onto specified faces of the body.
<code>project_curves</code>	Project all specified geometries onto the body.
<code>imprint_projected_curves</code>	Project and imprint specified geometries onto the body.
<code>set_name</code>	Set the name of the body.
<code>set_fill_style</code>	Set the fill style of the body.
<code>set_suppressed</code>	Set the body suppression state.
<code>set_color</code>	Set the color of the body.
<code>translate</code>	Translate the body in a specified direction and distance.
<code>rotate</code>	Rotate the geometry body around the specified axis by a given angle.
<code>scale</code>	Scale the geometry body by the given value.
<code>map</code>	Map the geometry body to the new specified frame.
<code>mirror</code>	Mirror the geometry body across the specified plane.
<code>get_collision</code>	Get the collision state between bodies.
<code>copy</code>	Create a copy of the body under the specified parent.
<code>get_named_selections</code>	Get the named selections associated with the body.
<code>get_raw_tessellation</code>	Tessellate the body and return the raw tessellation data.

continues on next page

Table 2 – continued from previous page

<i>tessellate</i>	Tessellate the body and return the geometry as triangles.
<i>get_vtk_tessellation</i>	Build VTK objects from raw tessellation data.
<i>shell_body</i>	Shell the body to the thickness specified.
<i>remove_faces</i>	Shell by removing a given set of faces.
<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.
<i>combine_merge</i>	Combine this body with another body or bodies, merging them into a single body.
<i>detach_faces</i>	Detach all the faces from the body.

Properties

<i>id</i>	Get the ID of the body as a string.
<i>name</i>	Get the name of the body.
<i>fill_style</i>	Get the fill style of the body.
<i>is_suppressed</i>	Get the body suppression state.
<i>color</i>	Get the color of the body.
<i>opacity</i>	Get the float value of the opacity for 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>vertices</i>	Get a list of all vertices 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>material</i>	Get the assigned material of the body.
<i>bounding_box</i>	Get the bounding box of the body.

Special methods

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

Import detail

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

Property detail

property Body.id: **str**

Get the ID of the body as a string.

property Body.name: **str**

Get the name of the body.

property Body.fill_style: **str**

Get the fill style of the body.

property `Body.is_suppressed: bool`

Get the body suppression state.

Warning

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

property `Body.color: str`

Get the color of the body.

property `Body.opacity: float`

Get the float value of the opacity for the body.

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

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

Get a list of all faces within the body.

Returns

`list[Face]`

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

Get a list of all edges within the body.

Returns

`list[Edge]`

property `Body.vertices: list[ansys.geometry.core.designer.vertex.Vertex]`

Get a list of all vertices within the body.

Returns

`list[Vertex]`

property `Body.is_alive: bool`

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

property `Body.is_surface: bool`

Check if the body is a planar body.

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

Get the surface thickness of a surface body.

Notes

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

property `Body.surface_offset: MidSurfaceOffsetType | None`

Get the surface offset type of a surface body.

Notes

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

property `Body.volume: pint.Quantity`

Calculate the volume of the body.

Notes

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

property `Body.material`: [`ansys.geometry.core.materials.material.Material`](#)

Get the assigned material of the body.

Returns

Material

Material assigned to the body.

property `Body.bounding_box`: [`ansys.geometry.core.math.bbox.BoundingBox`](#)

Get the bounding box of the body.

Returns

BoundingBox

Bounding box of the body.

Warning

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

Method detail

`Body.reset_tessellation_cache()`

Decorate Body methods that require a tessellation cache update.

Parameters

func

[method] Method to call.

Returns

Any

Output of the method, if any.

`Body.get_bounding_box(tight: bool = False) → ansys.geometry.core.math.bbox.BoundingBox`

Get the bounding box of the body.

Parameters

tight

[bool, default: False] Whether to use a tight tolerance when calculating the bounding box.

Returns

BoundingBox

Bounding box of the body.

Warning

This method is only available starting on Ansys release 27R1.

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

Assign a material against the active design.

Parameters**material**

[Material] Source material data.

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

Get the assigned material of the body.

Returns**Material**

Material assigned to the body.

Body.**remove_assigned_material()**

Remove the material assigned to the body.

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

Add a mid-surface thickness to a surface body.

Parameters**thickness**

[Distance | Quantity | Real] Thickness to assign.

Notes

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

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

Add a mid-surface offset to a surface body.

Parameters**offset_type**

[MidSurfaceOffsetType] Surface offset to assign.

Notes

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

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

Imprint all specified geometries onto specified faces of the body.

Parameters**faces: list[Face]**

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

sketch: Sketch

All curves to imprint on the faces.

Returns**tuple[list[Edge], list[Face]]**

All impacted edges and faces from the imprint operation.

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

Project all specified geometries onto the body.

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

list[Face]

All faces from the project curves operation.

Notes

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

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

Project and imprint specified geometries onto the body.

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

Parameters

direction: UnitVector3D

Direction of the projection.

sketch: Sketch

All curves to project on the body.

closest_face: bool

Whether to target the closest face with the projection.

only_one_curve: bool, default: False

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

Returns

list[Face]

All imprinted faces from the operation.

Notes

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

Body.**set_name**(*name: str*) → None

Set the name of the body.

Warning

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

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

Set the fill style of the body.

Warning

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

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

Set the body suppression state.

Warning

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

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

Set the color of the body.

Parameters

color

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

Warning

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

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

Translate the body in a specified direction and distance.

Parameters

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Distance (magnitude) of the translation.

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

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

Parameters

axis_origin: Point3D
Origin of the rotational axis.

axis_direction: UnitVector3D
The axis of rotation.

angle: ~pint.Quantity | Angle | Real
Angle (magnitude) of the rotation.

Warning

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

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

Scale the geometry body by the given value.

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

Parameters

value: Real
Value to scale the body by.

Warning

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

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

Map the geometry body to the new specified frame.

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

Parameters

frame: Frame
Structure defining the orientation of the body.

Warning

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

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

Mirror the geometry body across the specified plane.

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

Parameters

plane: Plane

Represents the mirror.

Warning

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

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

Get the collision state between bodies.

Parameters**body: Body**

Object that the collision state is checked with.

Returns**CollisionType**

Enum that defines the collision state between bodies.

Warning

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

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

Create a copy of the body under the specified parent.

Parameters**parent: Component**

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

name: str

Name to give the new body.

Returns**Body**

Copy of the body.

Body.**get_named_selections**() → *list[ansys.geometry.core.designer.selection.NamedSelection]*

Get the named selections associated with the body.

Returns**list[NamedSelection]**

List of named selections associated with the body.

Body.**get_raw_tessellation**(tess_options: *ansys.geometry.core.misc.options.TessellationOptions* | *None = None*, reset_cache: *bool = False*, include_faces: *bool = True*, include_edges: *bool = False*) → *dict*

Tessellate the body and return the raw tessellation data.

Parameters**transform**

[Matrix44, default: IDENTITY_MATRIX44] A transformation matrix to apply to the tessellation.

tess_options

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

reset_cache

[bool, default: **False**] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[bool, default: **True**] Whether to include face tessellation data in the output.

include_edges

[bool, default: **False**] Whether to include edge tessellation data in the output.

Returns

dict

Dictionary with face and edge IDs as keys and face and edge tessellation data as values.

Body.tessellate(merge: bool = False, tess_options: ansys.geometry.core.misc.options.TessellationOptions | None = None, reset_cache: bool = False, include_faces: bool = True, include_edges: bool = False) → pyvista.PolyData | pyvista.MultiBlock

Tessellate the body and return the geometry as triangles.

Parameters

merge

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

tess_options

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

reset_cache

[bool, default: **False**] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[bool, default: **True**] Whether to include face tessellation data in the output.

include_edges

[bool, default: **False**] Whether to include edge tessellation data in the output.

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])
```

(continues on next page)

(continued from previous page)

```

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

```

Merge the body:

```

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

```

Body.**get_vtk_tessellation**(*merge: bool = False, tess_options:*
ansys.geometry.core.misc.options.TessellationOptions | None = None, reset_cache:
bool = False, include_faces: bool = True, include_edges: bool = False,
_raw_tessellation: dict | None = None) →
vtkmodules.vtkCommonDataModel.vtkPolyData |
vtkmodules.vtkCommonDataModel.vtkMultiBlockDataSet

Build VTK objects from raw tessellation data.

Parameters

merge

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

tess_options

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

reset_cache

[*bool*, default: *False*] Whether to reset the tessellation cache and re-request the tessellation from the server.

include_faces

[*bool*, default: *True*] Whether to include face tessellation data in the output.

include_edges

[*bool*, default: *False*] Whether to include edge tessellation data in the output.

_raw_tessellation

[`dict` | `None`, default: `None`] Raw tessellation data with face and edge IDs as keys and tessellation data as values.. If `None`, the method requests the raw tessellation data from the server. This parameter is intended for internal use only.

Returns

`vtkPolyData`, `vtkMultiBlockDataSet`

Merged `vtkPolyData` if `merge=True` or a `vtkMultiBlockDataSet`.

Body.**`shell_body`**(*offset*: `ansys.geometry.core.misc.measurements.Distance` | `pint.Quantity` | `ansys.geometry.core.typing.Real`) → `bool`

Shell the body to the thickness specified.

Parameters

`offset`

[`Distance` | `Quantity` | `Real`] Shell thickness.

Returns

`bool`

True when successful, False when failed.

Warning

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

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

Shell by removing a given set of faces.

Parameters

`selection`

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

`offset`

[`Distance` | `Quantity` | `Real`] Shell thickness.

Returns

`bool`

True when successful, False when failed.

Warning

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

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

Plot the body.

Parameters

`merge`

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

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

screenshot

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

use_frame

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

use_service_colors

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

**plotting_options

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

Examples

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

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

Plot the body and color each face individually:

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

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

Intersect two (or more) bodies.

Parameters

other

[Body] Body to intersect with.

keep_other

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

Raises

ValueError

If the bodies do not intersect.

Notes

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

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

Subtract two (or more) bodies.

Parameters

`other`

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

`keep_other`

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

Raises

`ValueError`

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

Notes

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

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

Unite two (or more) bodies.

Parameters

`other`

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

`keep_other`

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

Notes

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

`Body.combine_merge(other: Body | list[Body]) → None`

Combine this body with another body or bodies, merging them into a single body.

Parameters

`other`

[Union[Body, list[Body]]] The body or list of bodies to combine with this body.

Notes

The `self` parameter is directly modified, and the `other` bodies are consumed.

`Body.detach_faces() → list[Body]`

Detach all the faces from the body.

This method will result in the original body becoming a surface body and a list of new surface bodies being created for every detached face.

Returns

`list[Body]`

Bodies created by the detach if any.

Warning

This method is only available starting on Ansys release 27R1.

`Body.__repr__() → str`

Represent the Body as a string.

MidSurfaceOffsetType

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

Bases: `enum.Enum`

Provides values for mid-surface offsets supported.

Overview

Attributes

<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`

FillStyle

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

Bases: `enum.Enum`

Provides values for fill styles supported.

Overview

Attributes

<i>DEFAULT</i>
<i>OPAQUE</i>
<i>TRANSPARENT</i>

Import detail

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

Attribute detail

`FillStyle.DEFAULT = 0`

`FillStyle.OPAQUE = 1`

`FillStyle.TRANSPARENT = 2`

Description

Provides for managing a body.

Module detail

`ansys.geometry.core.designer.body.__TEMPORARY_BOOL_OPS_FIX__ = (99, 0, 0)`

The `component.py` module

Summary

Classes

<i>SweepWithGuideData</i>	Data class for sweep with guide parameters.
<i>Component</i>	Provides for creating and managing a component.

Enums

<i>SharedTopologyType</i>	Shared topologies available.
<i>ExtrusionDirection</i>	Enum for extrusion direction definition.

`SweepWithGuideData`

class `ansys.geometry.core.designer.component.SweepWithGuideData`

Data class for sweep with guide parameters.

Parameters

- name**
[`str`] Name of the body to be generated by the sweep operation.
- parent_id**
[`str`] ID of the parent component.
- sketch**
[`Sketch`] Sketch to use for the sweep operation.
- path**
[`TrimmedCurve`] Path to sweep along.
- guide**
[`TrimmedCurve`] Guide curve for the sweep operation.
- tight_tolerance**
[`bool`] Whether to use tight tolerance for the sweep operation.

Overview

Attributes

<code>name</code>
<code>parent_id</code>
<code>sketch</code>
<code>path</code>
<code>guide</code>
<code>tight_tolerance</code>

Import detail

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

Attribute detail

`SweepWithGuideData.name: str`

`SweepWithGuideData.parent_id: str`

`SweepWithGuideData.sketch: ansys.geometry.core.sketch.sketch.Sketch`

`SweepWithGuideData.path: ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve`

`SweepWithGuideData.guide: ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve`

`SweepWithGuideData.tight_tolerance: bool = False`

Component

```
class ansys.geometry.core.designer.component.Component(name: str, parent_component: Component |  
    None, grpc_client: an-  
    sys.geometry.core.connection.client.GrpcClient,  
    template: Component | None = None,  
    instance_name: str | None = None,  
    preexisting_id: str | None = None,  
    master_component: an-  
    sys.geometry.core.designer.part.MasterComponent  
    | None = None, read_existing_comp: bool =  
    False)
```

Provides for creating and managing a component.

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

Parameters

name

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

parent_component

[`Component` or `None`] Parent component to place the new component under within the design assembly. The default is `None` only when dealing with a `Design` object.

grpc_client

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

template

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

instance_name: str, default: None

User defined optional name for the component instance.

preexisting_id

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

master_component

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

read_existing_comp

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

Overview

Methods

<code>set_name</code>	Set the name of the component.
<code>get_all_bodies</code>	Get all bodies in the component hierarchy.
<code>get_world_transform</code>	Get the full transformation matrix of the component in world space.
<code>modify_placement</code>	Apply a translation and/or rotation to the placement matrix.
<code>reset_placement</code>	Reset a component's placement matrix to an identity matrix.
<code>add_component</code>	Add a new component under this component within the design assembly.
<code>set_shared_topology</code>	Set the shared topology to apply to the component.
<code>extrude_sketch</code>	Create a solid body by extruding the sketch profile a distance.
<code>sweep_sketch</code>	Create a body by sweeping a planar profile along a path.
<code>sweep_chain</code>	Create a body by sweeping a chain of curves along a path.
<code>sweep_with_guide</code>	Create a body by sweeping a sketch along a path with a guide curve.
<code>revolve_sketch</code>	Create a solid body by revolving a sketch profile around an axis.
<code>extrude_face</code>	Extrude the face profile by a given distance to create a solid body.
<code>create_sphere</code>	Create a sphere body defined by the center point and the radius.
<code>create_body_from_loft_profile</code>	Create a lofted body from a collection of trimmed curves.
<code>create_body_from_loft_profiles_with_guides</code>	Create a lofted body from a collection of trimmed curves with guide curves.
<code>create_surface</code>	Create a surface body with a sketch profile.
<code>create_surface_from_face</code>	Create a surface body based on a face.

continues on next page

Table 3 – continued from previous page

<code>create_body_from_surface</code>	Create a surface body from a trimmed surface.
<code>create_surface_from_trimmed_curves</code>	Create a surface body from a list of trimmed curves all lying on the same plane.
<code>create_coordinate_system</code>	Create a coordinate system.
<code>translate_bodies</code>	Translate the bodies in a specified direction by a distance.
<code>create_beams</code>	Create beams under the component.
<code>create_beam</code>	Create a beam under the component.
<code>delete_component</code>	Delete a component (itself or its children).
<code>delete_body</code>	Delete a body belonging to this component (or its children).
<code>add_design_point</code>	Create a single design point.
<code>add_design_points</code>	Create a list of design points.
<code>create_datum_plane</code>	Create a datum plane on this component.
<code>delete_datum_plane</code>	Delete a datum plane from this component.
<code>delete_beam</code>	Delete an existing beam belonging to this component's scope.
<code>search_component</code>	Search nested components recursively for a component.
<code>search_body</code>	Search bodies in the component's scope.
<code>search_beam</code>	Search beams in the component's scope.
<code>search_plane</code>	Search planes in the component's scope.
<code>copy_faces</code>	Create a surface body from the faces provided.
<code>tessellate</code>	Tessellate the component.
<code>plot</code>	Plot the component.
<code>tree_print</code>	Print the component in tree format.
<code>import_named_selections</code>	Import named selections of a component.
<code>make_independent</code>	Make a component independent if it is an instance.
<code>get_named_selections</code>	Get the named selections of the component.

Properties

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

Special methods

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

Import detail

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

Property detail

property `Component.id: str`

ID of the component.

property `Component.name: str`

Name of the component.

property `Component.instance_name: str`

Name of the component instance.

Warning

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

property `Component.components: list[Component]`

List of Component objects inside of the component.

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

List of Body objects inside of the component.

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

List of Beam objects inside of the component.

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

List of DesignPoint objects inside of the component.

property `Component.datum_planes: list[ansys.geometry.core.designer.datumplane.DatumPlane]`

List of DatumPlane objects inside of the component.

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

List of CoordinateSystem objects inside of the component.

property `Component.parent_component: Component`

Parent of the component.

property `Component.is_alive: bool`

Whether the component is still alive on the server side.

property `Component.shared_topology: SharedTopologyType | None`

Shared topology type of the component (if any).

Notes

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

Method detail

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

Set the name of the component.

Warning

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

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

Get all bodies in the component hierarchy.

Returns

list[*Body*]

List of all bodies in the component hierarchy.

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

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

Returns

Matrix44

4x4 transformation matrix of the component in world space.

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

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

Parameters

translation

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

rotation_origin

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

rotation_direction

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

rotation_angle

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

Notes

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

Component.**reset_placement**()

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

See [modify_placement\(\)](#).

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

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

Parameters

name

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

template

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

Returns

Component

New component with no children in the design assembly.

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

Set the shared topology to apply to the component.

Parameters

share_type

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

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

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

Parameters

name

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

sketch

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

distance

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

direction

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

cut

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

Returns

Body

Extruded body from the given sketch.

None

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

Notes

The newly created body is placed under this component within the design assembly. Extruding a NURBS sketch requires a minimum Ansys release version of 26R1.

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

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

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

Parameters

name

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

sketch

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

path

[*list*[*TrimmedCurve*]] The path to sweep the profile along.

Returns

Body

Created body from the given sketch.

Warning

This method is only available starting on Ansys release 24R2. Sweeping a NURBS sketch requires a minimum Ansys release version of 26R1.

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

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

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

Parameters

name

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

path

[*list*[*TrimmedCurve*]] The path to sweep the chain along.

chain

[*list*[*TrimmedCurve*]] A chain of trimmed curves.

Returns

Body

Created body from the given sketch.

Warning

This method is only available starting on Ansys release 24R2. Sweeping NURBS curves requires a minimum Ansys release version of 26R1.

Component.**sweep_with_guide**(*sweep_data*: list[SweepWithGuideData]) →
list[ansys.geometry.core.designer.body.Body]

Create a body by sweeping a sketch along a path with a guide curve.

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

Parameters

sweep_data: list[SweepWithGuideData]

Data for the sweep operation, including the sketch, path, and guide curve.

Returns

list[Body]

Created bodies from the given sweep data.

Warning

This method is only available starting on Ansys release 26R1.

Component.**revolve_sketch**(*name*: str, *sketch*: ansys.geometry.core.sketch.sketch.Sketch, *axis*:
ansys.geometry.core.math.vector.Vector3D, *angle*: pint.Quantity |
ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real,
rotation_origin: ansys.geometry.core.math.point.Point3D) →
ansys.geometry.core.designer.body.Body

Create a solid body by revolving a sketch profile around an axis.

Parameters

name

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

sketch

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

axis

[Vector3D] Axis of rotation for the revolve.

angle

[Quantity | Angle | Real] Angle to revolve the solid body around the axis. The angle can be positive or negative.

rotation_origin

[Point3D] Origin of the axis of rotation.

Returns

Body

Revolved body from the given sketch.

Warning

This method is only available starting on Ansys release 24R2. Revolving a NURBS sketch requires a minimum Ansys release version of 26R1.

Component **.extrude_face**(*name: str*, *face: ansys.geometry.core.designer.face.Face*, *distance: pint.Quantity | ansys.geometry.core.misc.measurements.Distance*, *direction: ExtrusionDirection | str = ExtrusionDirection.POSITIVE*) → *ansys.geometry.core.designer.body.Body*

Extrude the face profile by a given distance to create a solid body.

There are no modifications against the body containing the source face.

Parameters**name**

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

face

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

distance

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

direction

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

Returns**Body**

Extruded solid body.

Notes

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

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

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

Parameters**name**

[*str*] Body name.

center

[*Point3D*] Center point of the sphere.

radius

[*Distance*] Radius of the sphere.

Returns**Body**

Sphere body object.

Warning

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

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

Create a lofted body from a collection of trimmed curves.

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

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

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

Parameters**name**

[*str*] Name of the lofted body.

profiles

[*list* [*list* [*TrimmedCurve*]]] Collection of lists of trimmed curves (profiles) defining the lofted body's shape.

periodic

[*bool*, default: *False*] Whether the lofted body should have periodic continuity.

ruled

[*bool*] Whether the lofted body should be ruled.

Returns**Body**

Created lofted body object.

Warning

This method is only available starting on Ansys release 24R2. Creating bodies from NURBS profiles requires a minimum Ansys release version of 26R1.

Component.**create_body_from_loft_profiles_with_guides**(*name: str, profiles: list[list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]], guides: list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]*) → *ansys.geometry.core.designer.body.Body*

Create a lofted body from a collection of trimmed curves with guide 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.

guides

[*list* [*TrimmedCurve*]] Collection of guide curves to influence the lofting process.

Returns

Body

Created lofted body object.

Warning

This method is only available starting on Ansys release 26R1.

`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.

Warning

Creating a surface from a NURBS sketch requires a minimum Ansys release version of 26R1.

`Component.create_surface_from_face(name: str, face: ansys.geometry.core.designer.face.Face) → ansys.geometry.core.designer.body.Body`

Create a surface body based on a face.

Parameters

name

[*str*] User-defined label for the new surface body.

face

[*Face*] Target face to use as the source for the new surface.

Returns

Body

Surface body.

Notes

The source face can be anywhere within the design component hierarchy. Therefore, there is no validation requiring that the face is placed under the target component where the body is to be created.

`Component.create_body_from_surface`(*name*: *str*, *trimmed_surface*: *ansys.geometry.core.shapes.surfaces.TrimmedSurface*) → *ansys.geometry.core.designer.body.Body*

Create a surface body from a trimmed surface.

It is possible to create a closed solid body (as opposed to an open surface body) with a Sphere or Torus if they are untrimmed. This can be validated with *body.is_surface*.

Parameters

name

[*str*] User-defined label for the new surface body.

trimmed_surface

[*TrimmedSurface*] Geometry for the new surface body.

Returns

Body

Surface body.

Warning

This method is only available starting on Ansys release 25R1. Creating a body from NURBS surfaces requires a minimum Ansys release version of 26R1.

`Component.create_surface_from_trimmed_curves`(*name*: *str*, *trimmed_curves*: *list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]*) → *ansys.geometry.core.designer.body.Body*

Create a surface body from a list of trimmed curves all lying on the same plane.

Parameters

name

[*str*] User-defined label for the new surface body.

trimmed_curves

[*list[TrimmedCurve]*] Curves to define the plane and body.

Returns

Body

Surface body.

Warning

This method is only available starting on Ansys release 24R2. Creating a surface from NURBS curves requires a minimum Ansys release version of 26R1.

`Component.create_coordinate_system`(*name*: *str*, *frame*: *ansys.geometry.core.math.frame.Frame*) → *ansys.geometry.core.designer.coordinate_system.CoordinateSystem*

Create a coordinate system.

The newly created coordinate system is place under this component within the design assembly.

Parameters

name

[*str*] User-defined label for the new coordinate system.

frame

[*Frame*] Frame defining the coordinate system bounds.

Returns

CoordinateSystem

`Component.translate_bodies(bodies: list[ansys.geometry.core.designer.body.Body], direction: ansys.geometry.core.math.vector.UnitVector3D, distance: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real) → None`

Translate the bodies in a specified direction by a distance.

Parameters

bodies: list[Body]

list of bodies to translate by the same distance.

direction: UnitVector3D

Direction of the translation.

distance: ~pint.Quantity | Distance | Real

Magnitude of the translation.

Notes

If the body does not belong to this component (or its children), it is not translated.

`Component.create_beams(segments: list[tuple[ansys.geometry.core.math.point.Point3D, ansys.geometry.core.math.point.Point3D]], profile: ansys.geometry.core.designer.beam.BeamProfile) → list[ansys.geometry.core.designer.beam.Beam]`

Create beams under the component.

Parameters

segments

[*list[tuple[Point3D, Point3D]]*] List of start and end pairs, each specifying a single line segment.

profile

[*BeamProfile*] Beam profile to use to create the beams.

Returns

list[Beam]

A list of the created Beams.

Notes

The newly created beams synchronize to a design within a supporting Geometry service instance.

`Component.create_beam(start: ansys.geometry.core.math.point.Point3D, end: ansys.geometry.core.math.point.Point3D, profile: ansys.geometry.core.designer.beam.BeamProfile) → ansys.geometry.core.designer.beam.Beam`

Create a beam under the component.

The newly created beam synchronizes to a design within a supporting Geometry service instance.

Parameters

start

[Point3D] Starting point of the beam line segment.

end

[Point3D] Ending point of the beam line segment.

profile

[BeamProfile] Beam profile to use to create the beam.

Component.**delete_component**(*component*: Component | *str*) → None

Delete a component (itself or its children).

Parameters

component

[Component | *str*] ID of the component or instance to delete.

Notes

If the component is not this component (or its children), it is not deleted.

Component.**delete_body**(*body*: ansys.geometry.core.designer.body.Body | *str*) → None

Delete a body belonging to this component (or its children).

Parameters

body

[Body | *str*] ID of the body or instance to delete.

Notes

If the body does not belong to this component (or its children), it is not deleted.

Component.**add_design_point**(*name*: *str*, *point*: ansys.geometry.core.math.point.Point3D) →
ansys.geometry.core.designer.designpoint.DesignPoint

Create a single design point.

Parameters

name

[*str*] User-defined label for the design points.

points

[Point3D] 3D point constituting the design point.

Component.**add_design_points**(*name*: *str*, *points*: list[ansys.geometry.core.math.point.Point3D]) →
list[ansys.geometry.core.designer.designpoint.DesignPoint]

Create a list of design points.

Parameters

name

[*str*] User-defined label for the list of design points.

points

[list[Point3D]] list of the 3D points that constitute the list of design points.

`Component.create_datum_plane(name: str, plane: ansys.geometry.core.math.plane.Plane) → ansys.geometry.core.designer.datumplane.DatumPlane`

Create a datum plane on this component.

Parameters

name

[str] User-defined label for the datum plane.

plane

[Plane] Plane object defining the datum plane's geometry.

Returns

DatumPlane

Created datum plane object.

Warning

This method is only available starting on Ansys release 27R1.

`Component.delete_datum_plane(plane: ansys.geometry.core.designer.datumplane.DatumPlane | str) → None`

Delete a datum plane from this component.

Parameters

plane

[DatumPlane | str] ID of the datum plane or instance to delete.

Notes

If the datum plane belongs to this component's children, it is deleted. If the datum plane does not belong to this component, it is not deleted.

`Component.delete_beam(beam: ansys.geometry.core.designer.beam.Beam | str) → None`

Delete an existing beam belonging to this component's scope.

Parameters

beam

[Beam | str] ID of the beam or instance to delete.

Notes

If the beam belongs to this component's children, it is deleted. If the beam does not belong to this component (or its children), it is not deleted.

`Component.search_component(id: str) → Component | None`

Search nested components recursively for a component.

Parameters

id

[str] ID of the component to search for.

Returns

Component

Component with the requested ID. If this ID is not found, None is returned.

Component.**search_body**(*id: str*) → *ansys.geometry.core.designer.body.Body* | *None*

Search bodies in the component's scope.

Parameters

id
[*str*] ID of the body to search for.

Returns

Body | **None**
Body with the requested ID. If the ID is not found, *None* is returned.

Notes

This method searches for bodies in the component and nested components recursively.

Component.**search_beam**(*id: str*) → *ansys.geometry.core.designer.beam.Beam* | *None*

Search beams in the component's scope.

Parameters

id
[*str*] ID of the beam to search for.

Returns

Beam | **None**
Beam with the requested ID. If the ID is not found, *None* is returned.

Notes

This method searches for beams in the component and nested components recursively.

Component.**search_plane**(*id: str*) → *ansys.geometry.core.designer.datumplane.DatumPlane* | *None*

Search planes in the component's scope.

Parameters

id
[*str*] ID of the plane to search for.

Returns

DatumPlane | **None**
DatumPlane with the requested ID. If the ID is not found, *None* is returned.

Notes

This method searches for planes in the component and nested components recursively.

Component.**copy_faces**(*name: str, faces: list[ansys.geometry.core.designer.face.Face]*) → *ansys.geometry.core.designer.body.Body*

Create a surface body from the faces provided.

Parameters

name
[*str*] Name of the new surface body.

faces
[*list*[*Face*]] List of faces to copy.

body

[Body] Body to which the faces belong.

Returns

Body

New surface body created from the copied faces.

`Component.tessellate(tess_options: ansys.geometry.core.misc.options.TessellationOptions | None = None,
_recursive_call: bool = False)` → `pyvista.PolyData` | `list[pyvista.MultiBlock]`

Tessellate the component.

Parameters

tess_options

[[TessellationOptions](#) | *None*, default: *None*] A set of options to determine the tessellation quality.

_recursive_call: bool, default: False

Internal flag to indicate if this method is being called recursively. Not to be used by the user.

Returns

[PolyData](#), [list\[MultiBlock\]](#)

Tessellated component as a single `PolyData` object. If the method is called recursively, a list of `MultiBlock` objects is returned.

`Component.plot(merge_component: bool = True, merge_bodies: bool = True, screenshot: str | None = None,
use_trame: bool | None = None, use_service_colors: bool | None = None, allow_picking: bool |
None = None, **plotting_options: dict | None)` → *None* | `list[Any]`

Plot the component.

Parameters

merge_component

[*bool*, default: *True*] Whether to merge the component into a single dataset. By default, *True*. Performance improved. When *True*, all the faces of the component are effectively merged into a single dataset. If *False*, the individual bodies are kept separate.

merge_bodies

[*bool*, default: *True*] Whether to merge each body into a single dataset. By default, *True*. Performance improved. When *True*, all the faces of each individual body are effectively merged into a single dataset. If *False*, the individual faces are kept separate.

screenshot

[*str*, default: *None*] Path for saving a screenshot of the image being represented.

use_trame

[*bool*, default: *None*] Whether to enable the use of `trame`. The default is *None*, in which case the `ansys.tools.visualization_interface.USE_TRAME` global setting is used.

use_service_colors

[*bool*, default: *None*] Whether to use the colors assigned to the body in the service. The default is *None*, in which case the `ansys.geometry.core.USE_SERVICE_COLORS` global setting is used.

allow_picking

[*bool*, default: *None*] Whether to enable picking. The default is *None*, in which case the picker is not enabled.

****plotting_options**

[dict, default: None] Keyword arguments for plotting. For allowable keyword arguments, see the

Returns

None | list[Any]

If allow_picking=True, a list of picked objects is returned. Otherwise, None.

Examples

Create 25 small cylinders in a grid-like pattern on the XY plane and plot them. Make the cylinders look metallic by enabling physically-based rendering with pbr=True.

```
>>> from ansys.geometry.core.misc.units import UNITS as u
>>> from ansys.geometry.core.sketch import Sketch
>>> from ansys.geometry.core.math import Plane, Point2D, Point3D, UnitVector3D
>>> from ansys.geometry.core import Modeler
>>> import numpy as np
>>> modeler = Modeler()
>>> origin = Point3D([0, 0, 0])
>>> plane = Plane(origin, direction_x=[1, 0, 0], direction_y=[0, 1, 0])
>>> design = modeler.create_design("my-design")
>>> mycomp = design.add_component("my-comp")
>>> n = 5
>>> xx, yy = np.meshgrid(
...     np.linspace(-4, 4, n),
...     np.linspace(-4, 4, n),
... )
>>> for x, y in zip(xx.ravel(), yy.ravel()):
...     sketch = Sketch(plane)
...     sketch.circle(Point2D([x, y]), 0.2 * u.m)
...     mycomp.extrude_sketch(f"body-{x}-{y}", sketch, 1 * u.m)
>>> mycomp
ansys.geometry.core.designer.Component 0x2203cc9ec50
  Name          : my-comp
  Exists        : True
  Parent component : my-design
  N Bodies      : 25
  N Components   : 0
  N Coordinate Systems : 0
>>> mycomp.plot(pbr=True, metallic=1.0)
```

Component.__repr__() → str

Represent the Component as a string.

Component.tree_print(consider_comps: bool = True, consider_bodies: bool = True, consider_beams: bool = True, depth: int | None = None, indent: int = 4, sort_keys: bool = False, return_list: bool = False, skip_loc_header: bool = False) → None | list[str]

Print the component in tree format.

Parameters**consider_comps**

[bool, default: True] Whether to print the nested components.

consider_bodies

[bool, default: True] Whether to print the bodies.

consider_beams

[bool, default: True] Whether to print the beams.

depth

[int | None, default: None] Depth level to print. If None, it prints all levels.

indent

[int, default: 4] Indentation level. Minimum is 2 - if less than 2, it is set to 2 by default.

sort_keys

[bool, default: False] Whether to sort the keys alphabetically.

return_list

[bool, default: False] Whether to return a list of strings or print out the tree structure.

skip_loc_header

[bool, default: False] Whether to skip the location header. Mostly for internal use.

Returns

None | list[str]

Tree-style printed component or list of strings representing the component tree.

Component.**import_named_selections()** → None

Import named selections of a component.

When a design is inserted, it becomes a component. From 26R1 onwards, the named selections of that component will be imported by default. If a file is opened that contains a component that did not have its named selections imported, this method can be used to import them.

Warning

This method is only available starting on Ansys release 26R1.

Component.**make_independent**(others: list[Component] = None) → None

Make a component independent if it is an instance.

If a component is an instance of another component, modifying one component modifies both. When a component is made independent, it is no longer associated with other instances and can be modified separately.

Parameters**others**

[list[Component], default: None] Optionally include multiple components to make them all independent.

Warning

This method is only available starting on Ansys release 26R1.

Component.**get_named_selections()** → list[ansys.geometry.core.designer.selection.NamedSelection]

Get the named selections of the component.

Returns

`list[NamedSelection]`

List of named selections belonging to the component.

SharedTopologyType

`class ansys.geometry.core.designer.component.SharedTopologyType(*args, **kws)`

Bases: `enum.Enum`

Shared topologies available.

Overview

Attributes

<code>SHARETYPE_NONE</code>
<code>SHARETYPE_SHARE</code>
<code>SHARETYPE_MERGE</code>
<code>SHARETYPE_GROUPS</code>

Import detail

```
from ansys.geometry.core.designer.component import SharedTopologyType
```

Attribute detail

`SharedTopologyType.SHARETYPE_NONE = 0`

`SharedTopologyType.SHARETYPE_SHARE = 1`

`SharedTopologyType.SHARETYPE_MERGE = 2`

`SharedTopologyType.SHARETYPE_GROUPS = 3`

ExtrusionDirection

`class ansys.geometry.core.designer.component.ExtrusionDirection(*args, **kws)`

Bases: `enum.Enum`

Enum for extrusion direction definition.

Overview

Constructors

<code>from_string</code> Convert a string to an ExtrusionDirection enum.
--

Methods

<code>get_multiplier</code> Get the multiplier for the extrusion direction.

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.

ExtrusionDirection.**get_multiplier**() → *int*

Get the multiplier for the extrusion direction.

Returns

int

1 if the direction is positive, -1 if negative.

Description

Provides for managing components.

The coordinate_system.py module

Summary

Classes

<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: str |
None = None)
```

Represents a user-defined coordinate system within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

name

[*str*] User-defined label for the coordinate system.

frame

[Frame] Frame defining the coordinate system bounds.

parent_component

[Component, default: Component] Parent component the coordinate system is assigned against.

grpc_client

[GrpcClient] Active supporting Geometry service instance for design modeling.

Overview

Properties

<i>id</i>	ID of the coordinate system.
<i>name</i>	Name of the coordinate system.
<i>frame</i>	Frame of the coordinate system.
<i>parent_component</i>	Parent component of the coordinate system.
<i>is_alive</i>	Flag indicating if coordinate system is still alive on the server.

Special methods

<i>__repr__</i>	Represent the coordinate system as a string.
-----------------	--

Import detail

```
from ansys.geometry.core.designer.coordinate_system import CoordinateSystem
```

Property detail

property `CoordinateSystem.id: str`

ID of the coordinate system.

property `CoordinateSystem.name: str`

Name of the coordinate system.

property `CoordinateSystem.frame: ansys.geometry.core.math.frame.Frame`

Frame of the coordinate system.

property `CoordinateSystem.parent_component:`
`ansys.geometry.core.designer.component.Component`

Parent component of the coordinate system.

property `CoordinateSystem.is_alive: bool`

Flag indicating if coordinate system is still alive on the server.

Method detail

`CoordinateSystem.__repr__() → str`

Represent the coordinate system as a string.

Description

Provides for managing a user-defined coordinate system.

The `datumplane.py` module

Summary

Classes

<i>DatumPlane</i>	Provides for creating datum planes in components.
-------------------	---

DatumPlane

```
class ansys.geometry.core.designer.datumplane.DatumPlane(id: str, name: str, plane:
                                                         ansys.geometry.core.math.plane.Plane,
                                                         parent_component: an-
                                                         sys.geometry.core.designer.component.Component)
```

Provides for creating datum planes in components.

Parameters

id

[str] Server-defined ID for the datum plane.

name

[str] User-defined label for the datum plane.

plane

[Plane] Plane constituting the datum plane.

parent_component

[Component] Parent component to place the new datum plane under within the design assembly.

Overview

Methods

<i>evaluate</i>	Evaluate the plane at UV parametric coordinates.
-----------------	--

Properties

<i>id</i>	ID of the datum plane.
<i>name</i>	Name of the datum plane.
<i>value</i>	Plane constituting the datum plane.
<i>parent_component</i>	Parent component of the datum plane.
<i>is_alive</i>	Check if the datum plane is still present on the server.

Special methods

<code>__repr__</code>	Represent the datum plane as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.designer.datumplane import DatumPlane
```

Property detail

property `DatumPlane.id: str`

ID of the datum plane.

property `DatumPlane.name: str`

Name of the datum plane.

property `DatumPlane.value: ansys.geometry.core.math.plane.Plane`

Plane constituting the datum plane.

property `DatumPlane.parent_component: ansys.geometry.core.designer.component.Component`

Parent component of the datum plane.

property `DatumPlane.is_alive: bool`

Check if the datum plane is still present on the server.

Method detail

`DatumPlane.evaluate(u: float, v: float) → ansys.geometry.core.math.point.Point3D`

Evaluate the plane at UV parametric coordinates.

This method converts 2D parametric coordinates (u, v) to a 3D Cartesian point on the plane using the parametric equation: $P(u, v) = \text{origin} + u * \text{direction_x} + v * \text{direction_y}$

Parameters

u

[float] First parametric coordinate along the plane's x-direction.

v

[float] Second parametric coordinate along the plane's y-direction.

Returns

Point3D

3D Cartesian point on the plane at the given UV coordinates.

`DatumPlane.__repr__()`

Represent the datum plane as a string.

Description

Module for creating and managing datum planes.

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.
-------------------------	---

Constants

<i>DESIGN_FILE_FORMAT_EXTENSIONS</i>

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

<code>close</code>	Close the design.
<code>add_material</code>	Add a material to the design.
<code>remove_material</code>	Remove a material from the design.
<code>save</code>	Save a design to disk on the active Geometry server instance.
<code>download</code>	Export and download the design from the server.
<code>export_to_scdocx</code>	Export the design to an scdocx file.
<code>export_to_disco</code>	Export the design to a DISCO file.
<code>export_to_stride</code>	Export the design to a stride file.
<code>export_to_parasolid_text</code>	Export the design to a Parasolid text file.
<code>export_to_parasolid_bin</code>	Export the design to a Parasolid binary file.
<code>export_to_fmd</code>	Export the design to an FMD file.
<code>export_to_step</code>	Export the design to a STEP file.
<code>export_to_iges</code>	Export the design to an IGES file.
<code>export_to_pmdb</code>	Export the design to a PMDB file.
<code>create_named_selection</code>	Create a named selection on the active Geometry server instance.
<code>delete_named_selection</code>	Delete a named selection on the active Geometry server instance.
<code>delete_component</code>	Delete a component (itself or its children).
<code>set_shared_topology</code>	Set the shared topology to apply to the component.
<code>add_beam_circular_profile</code>	Add a new beam circular profile under the design for creating beams.
<code>get_all_parameters</code>	Get parameters for the design.
<code>set_parameter</code>	Set or update a parameter of the design.
<code>add_midsurface_thickness</code>	Add a mid-surface thickness to a list of bodies.
<code>add_midsurface_offset</code>	Add a mid-surface offset type to a list of bodies.
<code>delete_beam_profile</code>	Remove a beam profile on the active geometry server instance.
<code>insert_file</code>	Insert a file into the design.
<code>get_raw_tessellation</code>	Tessellate the entire design and return the geometry as triangles.

Properties

<code>design_id</code>	The design's object unique id.
<code>materials</code>	List of materials available for the design.
<code>named_selections</code>	List of named selections available for the design.
<code>beam_profiles</code>	List of beam profile available for the design.
<code>parameters</code>	List of parameters available for the design.
<code>is_active</code>	Whether the design is currently active.
<code>is_closed</code>	Whether the design is closed (i.e. not active).

Special methods

<code>__repr__</code>	Represent the Design as a string.
-----------------------	-----------------------------------

Import detail

```
from ansys.geometry.core.designer.design import Design
```


Property detail

property `Design.design_id: str`

The design's object unique id.

property `Design.materials: list[ansys.geometry.core.materials.material.Material]`

List of materials available for the design.

property `Design.named_selections: list[ansys.geometry.core.designer.selection.NamedSelection]`

List of named selections available for the design.

property `Design.beam_profiles: list[ansys.geometry.core.designer.beam.BeamProfile]`

List of beam profile available for the design.

property `Design.parameters: list[ansys.geometry.core.parameters.parameter.Parameter]`

List of parameters available for the design.

property `Design.is_active: bool`

Whether the design is currently active.

property `Design.is_closed: bool`

Whether the design is closed (i.e. not active).

Method detail

`Design.close() → None`

Close the design.

`Design.add_material(material: ansys.geometry.core.materials.material.Material) → None`

Add a material to the design.

Parameters

material

[Material] Material to add.

`Design.remove_material(material: ansys.geometry.core.materials.material.Material | list[ansys.geometry.core.materials.material.Material]) → None`

Remove a material from the design.

Parameters

material

[Material | list[Material]] Material or list of materials to remove.

`Design.save(file_location: pathlib.Path | str, write_body_facets: bool = False) → None`

Save a design to disk on the active Geometry server instance.

Parameters

file_location

[Path | str] Location on disk to save the file to.

write_body_facets

[bool, default: False] Option to write body facets into the saved file. 26R1 and later.

`Design.download(file_location: pathlib.Path | str, format: DesignFileFormat = DesignFileFormat.SCDOCX, write_body_facets: bool = False) → None`

Export and download the design from the server.

Parameters

`file_location`

[*Path* | *str*] Location on disk to save the file to.

`format`

[*DesignFileFormat*, default: *DesignFileFormat.SCDOCX*] Format for the file to save to.

`write_body_facets`

[*bool*, default: *False*] Option to write body facets into the saved file. SCDOCX and DISCO only, 26R1 and later.

`Design.export_to_scdocx(location: pathlib.Path | str | None = None, write_body_facets: bool = False) → pathlib.Path`

Export the design to an scdocx file.

Parameters

`location`

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

`write_body_facets`

[*bool*, default: *False*] Option to write body facets into the saved file. SCDOCX and DISCO only, 26R1 and later.

Returns

Path

The path to the saved file.

`Design.export_to_disco(location: pathlib.Path | str | None = None, write_body_facets: bool = False) → pathlib.Path`

Export the design to a DISCO file.

Parameters

`location`

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

`write_body_facets`

[*bool*, default: *False*] Option to write body facets into the saved file. SCDOCX and DISCO only, 26R1 and later.

Returns

Path

The path to the saved file.

`Design.export_to_stride(location: pathlib.Path | str | None = None) → pathlib.Path`

Export the design to a stride file.

Parameters

`location`

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

`Design.export_to_parasolid_text(location: pathlib.Path | str | None = None) → pathlib.Path`

Export the design to a Parasolid text file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

`Design.export_to_parasolid_bin(location: pathlib.Path | str | None = None) → pathlib.Path`

Export the design to a Parasolid binary file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

`Design.export_to_fmd(location: pathlib.Path | str | None = None) → pathlib.Path`

Export the design to an FMD file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

`Design.export_to_step(location: pathlib.Path | str | None = None) → pathlib.Path`

Export the design to a STEP file.

Parameters**location**

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns**Path**

The path to the saved file.

`Design.export_to_iges(location: pathlib.Path | str = None) → pathlib.Path`

Export the design to an IGES file.

Parameters

location

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns

Path

The path to the saved file.

`Design.export_to_pmdb(location: pathlib.Path | str | None = None) → pathlib.Path`

Export the design to a PMDB file.

Parameters

location

[*Path* | *str*, optional] Location on disk to save the file to. If *None*, the file will be saved in the current working directory.

Returns

Path

The path to the saved file.

`Design.create_named_selection(name: str, bodies: list[ansys.geometry.core.designer.body.Body] | None = None, faces: list[ansys.geometry.core.designer.face.Face] | None = None, edges: list[ansys.geometry.core.designer.edge.Edge] | None = None, beams: list[ansys.geometry.core.designer.beam.Beam] | None = None, design_points: list[ansys.geometry.core.designer.designpoint.DesignPoint] | None = None, components: list[ansys.geometry.core.designer.component.Component] | None = None, vertices: list[ansys.geometry.core.designer.vertex.Vertex] | None = None) → ansys.geometry.core.designer.selection.NamedSelection`

Create a named selection on the active Geometry server instance.

Parameters

name

[*str*] User-defined name for the named selection.

bodies

[*list*[*Body*], default: *None*] All bodies to include in the named selection.

faces

[*list*[*Face*], default: *None*] All faces to include in the named selection.

edges

[*list*[*Edge*], default: *None*] All edges to include in the named selection.

beams

[*list*[*Beam*], default: *None*] All beams to include in the named selection.

design_points

[*list*[*DesignPoint*], default: *None*] All design points to include in the named selection.

components

[*list*[*Component*], default: *None*] All components to include in the named selection.

vertices

[*list*[*Vertex*], default: *None*] All vertices to include in the named selection.

Returns**NamedSelection**

Newly created named selection that maintains references to all target entities.

Raises**ValueError**

If no entities are provided for the named selection. At least one of the optional parameters must be provided.

`Design.delete_named_selection(named_selection: ansys.geometry.core.designer.selection.NamedSelection | str) → None`

Delete a named selection on the active Geometry server instance.

Parameters**named_selection**

[NamedSelection | str] Name of the named selection or instance.

`Design.delete_component(component: ansys.geometry.core.designer.component.Component | str) → None`

Delete a component (itself or its children).

Parameters**id**

[Union[Component, str]] Name of the component or instance to delete.

Raises**ValueError**

The design itself cannot be deleted.

Notes

If the component is not this component (or its children), it is not deleted.

`Design.set_shared_topology(share_type: ansys.geometry.core.designer.component.SharedTopologyType) → None`

Set the shared topology to apply to the component.

Parameters**share_type**

[SharedTopologyType] Shared topology type to assign.

Raises**ValueError**

Shared topology does not apply to a design.

`Design.add_beam_circular_profile(name: str, radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance, center: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D = ZERO_POINT3D, direction_x: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = UNITVECTOR3D_X, direction_y: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.vector.UnitVector3D | ansys.geometry.core.math.vector.Vector3D = UNITVECTOR3D_Y) → ansys.geometry.core.designer.beam.BeamCircularProfile`

Add a new beam circular profile under the design for creating beams.

Parameters

name

[[str](#)] User-defined label for the new beam circular profile.

radius

[[Quantity](#) | [Distance](#)] Radius of the beam circular profile.

center

[[ndarray](#) | [RealSequence](#) | [Point3D](#)] Center of the beam circular profile.

direction_x

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] X-plane direction.

direction_y

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] Y-plane direction.

`Design.get_all_parameters()` → [list\[ansys.geometry.core.parameters.parameter.Parameter\]](#)

Get parameters for the design.

Returns

[list\[Parameter\]](#)

List of parameters for the design.

Warning

This method is only available starting on Ansys release 25R1.

`Design.set_parameter(dimension: ansys.geometry.core.parameters.parameter.Parameter)` → [ansys.geometry.core.parameters.parameter.ParameterUpdateStatus](#)

Set or update a parameter of the design.

Parameters

dimension

[[Parameter](#)] Parameter to set.

Returns

[ParameterUpdateStatus](#)

Status of the update operation.

Warning

This method is only available starting on Ansys release 25R1.

`Design.add_midsurface_thickness(thickness: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real, bodies: list\[ansys.geometry.core.designer.body.Body\])` → [None](#)

Add a mid-surface thickness to a list of bodies.

Parameters

thickness

[[Quantity](#)] Thickness to be assigned.

bodies

[list[Body]] All bodies to include in the mid-surface thickness assignment.

Notes

Only surface bodies will be eligible for mid-surface thickness assignment.

`Design.add_midsurface_offset(offset_type: ansys.geometry.core.designer.body.MidSurfaceOffsetType, bodies: list[ansys.geometry.core.designer.body.Body]) → None`

Add a mid-surface offset type to a list of bodies.

Parameters

offset_type

[MidSurfaceOffsetType] Surface offset to be assigned.

bodies

[list[Body]] All bodies to include in the mid-surface offset assignment.

Notes

Only surface bodies will be eligible for mid-surface offset assignment.

`Design.delete_beam_profile(beam_profile: ansys.geometry.core.designer.beam.BeamProfile | str) → None`

Remove a beam profile on the active geometry server instance.

Parameters

beam_profile

[BeamProfile | str] A beam profile name or instance that should be deleted.

`Design.insert_file(file_location: pathlib.Path | str, import_options: ansys.geometry.core.misc.options.ImportOptions = ImportOptions(), import_options_definitions: ansys.geometry.core.misc.options.ImportOptionsDefinitions = ImportOptionsDefinitions()) → ansys.geometry.core.designer.component.Component`

Insert a file into the design.

Parameters

file_location

[Path | str] Location on disk where the file is located.

import_options

[ImportOptions, optional] The options to pass into upload file. If none are provided, default options are used.

import_options_definitions

[ImportOptionsDefinitions, optional] Additional options to pass into insert file. If none are provided, default options are used.

Returns

Component

The newly inserted component.

Warning

This method is only available starting on Ansys release 24R2.

Design.get_raw_tessellation(*tess_options*: `ansys.geometry.core.misc.options.TessellationOptions` | *None* = *None*, *reset_cache*: *bool* = *False*, *include_faces*: *bool* = *True*, *include_edges*: *bool* = *False*) → *dict*

Tessellate the entire design and return the geometry as triangles.

Parameters

tess_options

[`TessellationOptions`, optional] Options for the tessellation. If *None*, default options are used.

reset_cache

[*bool*, default: *False*] Whether to reset the cache before performing the tessellation.

include_faces

[*bool*, default: *True*] Whether to include faces in the tessellation.

include_edges

[*bool*, default: *False*] Whether to include edges in the tessellation.

Returns

dict

A dictionary with body IDs as keys and another dictionary as values. The inner dictionary has face and edge IDs as keys and the corresponding face/vertice arrays as values.

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>SCDOX</i>
<i>PARASOLID_TEXT</i>
<i>PARASOLID_BIN</i>
<i>FMD</i>
<i>STEP</i>
<i>IGES</i>
<i>PMDB</i>
<i>STRIDE</i>
<i>DISCO</i>
<i>INVALID</i>

Special methods

<code>__str__</code>	Represent object in string format.
----------------------	------------------------------------

Import detail

```
from ansys.geometry.core.designer.design import DesignFileFormat
```

Attribute detail

```
DesignFileFormat.SDOCKX = 'SDOCKX'  
DesignFileFormat.PARASOLID_TEXT = 'PARASOLID_TEXT'  
DesignFileFormat.PARASOLID_BIN = 'PARASOLID_BIN'  
DesignFileFormat.FMD = 'FMD'  
DesignFileFormat.STEP = 'STEP'  
DesignFileFormat.IGES = 'IGES'  
DesignFileFormat.PMDB = 'PMDB'  
DesignFileFormat.STRIDE = 'STRIDE'  
DesignFileFormat.DISCO = 'DISCO'  
DesignFileFormat.INVALID = 'INVALID'
```

Method detail

```
DesignFileFormat.__str__()
    Represent object in string format.
```

Description

Provides for managing designs.

Module detail

```
ansys.geometry.core.designer.design.DESIGN_FILE_FORMAT_EXTENSIONS
```

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: an-  
sys.geometry.core.math.point.Point3D,  
parent_component: an-  
sys.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

Methods

<code>get_named_selections</code>	Get named selections that contain this design point.
-----------------------------------	--

Properties

<code>id</code>	ID of the design point.
<code>name</code>	Name of the design point.
<code>value</code>	Value of the design point.
<code>parent_component</code>	Component node that the design point is under.

Special methods

<code>__repr__</code>	Represent the design points as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.designer.designpoint import DesignPoint
```

Property detail

property DesignPoint.id: str

ID of the design point.

property DesignPoint.name: str

Name of the design point.

property DesignPoint.value: `ansys.geometry.core.math.point.Point3D`

Value of the design point.

property DesignPoint.parent_component: `ansys.geometry.core.designer.component.Component`

Component node that the design point is under.

Method detail

`DesignPoint.get_named_selections()` → `list[ansys.geometry.core.designer.selection.NamedSelection]`

Get named selections that contain this design point.

Returns

`list[NamedSelection]`

List of named selections that contain this design point.

`DesignPoint.__repr__()` → `str`

Represent the design points as a string.

Description

Module for creating and managing design points.

The `edge.py` module

Summary

Classes

<code>Edge</code> Represents a single edge of a body within the design assembly.
--

Enums

<code>CurveType</code> Provides values for the curve types supported.

Edge

class `ansys.geometry.core.designer.edge.Edge`(*id*: `str`, *curve_type*: `CurveType`, *body*: `ansys.geometry.core.designer.body.Body`, *grpc_client*: `ansys.geometry.core.connection.client.GrpcClient`, *is_reversed*: `bool = False`)

Represents a single edge of a body within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

id

[`str`] Server-defined ID for the body.

curve_type

[`CurveType`] Type of curve that the edge forms.

body

[`Body`] Parent body that the edge constructs.

grpc_client

[`GrpcClient`] Active supporting Geometry service instance for design modeling.

is_reversed

[`bool`] Direction of the edge.

Overview

Methods

<code>get_bounding_box</code>	Get the tight bounding box for the face.
<code>get_named_selections</code>	Get named selections associated with the edge.

Properties

<code>id</code>	ID of the edge.
<code>body</code>	Body of the edge.
<code>is_reversed</code>	Flag indicating if the edge is reversed.
<code>shape</code>	Underlying trimmed curve of the edge.
<code>length</code>	Calculated length of the edge.
<code>curve_type</code>	Curve type of the edge.
<code>faces</code>	Faces that contain the edge.
<code>vertices</code>	Vertices that define the edge.
<code>start</code>	Start point of the edge.
<code>end</code>	End point of the edge.
<code>bounding_box</code>	Bounding box of the edge.

Import detail

```
from ansys.geometry.core.designer.edge import Edge
```

Property detail

property `Edge.id`: `str`

ID of the edge.

property `Edge.body`: `ansys.geometry.core.designer.body.Body`

Body of the edge.

property `Edge.is_reversed`: `bool`

Flag indicating if the edge is reversed.

property `Edge.shape`: `ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve`

Underlying trimmed curve of the edge.

If the edge is reversed, its shape is the `ReversedTrimmedCurve` type, which swaps the start and end points of the curve and handles parameters to allow evaluation as if the curve is not reversed.

Warning

This method is only available starting on Ansys release 24R2.

property `Edge.length`: `pint.Quantity`

Calculated length of the edge.

property `Edge.curve_type`: `CurveType`

Curve type of the edge.

property Edge.faces: `list[ansys.geometry.core.designer.face.Face]`

Faces that contain the edge.

property Edge.vertices: `list[ansys.geometry.core.designer.vertex.Vertex]`

Vertices that define the edge.

property Edge.start: `ansys.geometry.core.math.point.Point3D`

Start point of the edge.

property Edge.end: `ansys.geometry.core.math.point.Point3D`

End point of the edge.

property Edge.bounding_box: `ansys.geometry.core.math.bbox.BoundingBox`

Bounding box of the edge.

Warning

This method is only available starting on Ansys release 25R2.

Method detail

Edge.get_bounding_box(*tight: bool = False*) → `ansys.geometry.core.math.bbox.BoundingBox`

Get the tight bounding box for the face.

Warning

This method is only available starting on Ansys release 27R1.

Edge.get_named_selections() → `list[ansys.geometry.core.designer.selection.NamedSelection]`

Get named selections associated with the edge.

Returns

`list[NamedSelection]`

List of named selections that include the edge.

CurveType

class ansys.geometry.core.designer.edge.CurveType(*args, **kwargs)

Bases: `enum.Enum`

Provides values for the curve types supported.

Overview

Attributes

CURVETYPE_UNKNOWN
CURVETYPE_LINE
CURVETYPE_CIRCLE
CURVETYPE_ELLIPSE
CURVETYPE_NURBS
CURVETYPE_PROCEDURAL

Import detail

```
from ansys.geometry.core.designer.edge import CurveType
```

Attribute detail

`CurveType.CURVETYPE_UNKNOWN = 0`

`CurveType.CURVETYPE_LINE = 1`

`CurveType.CURVETYPE_CIRCLE = 2`

`CurveType.CURVETYPE_ELLIPSE = 3`

`CurveType.CURVETYPE_NURBS = 4`

`CurveType.CURVETYPE_PROCEDURAL = 5`

Description

Module for managing an edge.

The face.py module

Summary

Classes

<i>FaceLoop</i>	Provides an internal class holding the face loops defined.
<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.
<i>FaceLoopType</i>	Provides values for the face loop types supported.

FaceLoop

```
class ansys.geometry.core.designer.face.FaceLoop(type: FaceLoopType, length: pint.Quantity,  
                                                min_bbox: ansys.geometry.core.math.point.Point3D,  
                                                max_bbox: ansys.geometry.core.math.point.Point3D,  
                                                edges:  
                                                list[ansys.geometry.core.designer.edge.Edge])
```

Provides an internal class holding the face loops defined.

Parameters

type

[FaceLoopType] Type of loop.

length

[Quantity] Length of the loop.

min_bbox

[Point3D] Minimum point of the bounding box containing the loop.

max_bbox

[Point3D] Maximum point of the bounding box containing the loop.

edges

[list[Edge]] Edges contained in the loop.

Notes

This class is to be used only when parsing server side results. It is not intended to be instantiated by a user.

Overview**Properties**

<i>type</i>	Type of the loop.
<i>length</i>	Length of the loop.
<i>min_bbox</i>	Minimum point of the bounding box containing the loop.
<i>max_bbox</i>	Maximum point of the bounding box containing the loop.
<i>edges</i>	Edges contained in the loop.

Import detail

```
from ansys.geometry.core.designer.face import FaceLoop
```

Property detail

property FaceLoop.type: *FaceLoopType*

Type of the loop.

property FaceLoop.length: *pint.Quantity*

Length of the loop.

property FaceLoop.min_bbox: *ansys.geometry.core.math.point.Point3D*

Minimum point of the bounding box containing the loop.

property FaceLoop.max_bbox: *ansys.geometry.core.math.point.Point3D*

Maximum point of the bounding box containing the loop.

property FaceLoop.edges: *list[ansys.geometry.core.designer.edge.Edge]*

Edges contained in the loop.

Face

```
class ansys.geometry.core.designer.face.Face(id: str, surface_type: SurfaceType, body:
    ansys.geometry.core.designer.body.Body, grpc_client:
    ansys.geometry.core.connection.client.GrpcClient,
    is_reversed: bool = False)
```

Represents a single face of a body within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

id
[str] Server-defined ID for the body.

surface_type
[SurfaceType] Type of surface that the face forms.

body
[Body] Parent body that the face constructs.

grpc_client
[GrpcClient] Active supporting Geometry service instance for design modeling.

Overview

Methods

<i>get_bounding_box</i>	Get the tight bounding box for the face.
<i>set_color</i>	Set the color of the face.
<i>set_opacity</i>	Set the opacity of the face.
<i>normal</i>	Get the normal direction to the face at certain UV coordinates.
<i>point</i>	Get a point of the face evaluated at certain UV coordinates.
<i>create_isoparametric_curves</i>	Create isoparametric curves at the given proportional parameter.
<i>setup_offset_relationship</i>	Create an offset relationship between two faces.
<i>get_named_selections</i>	Get named selections associated with the edge.
<i>tessellate</i>	Tessellate the face and return the geometry as triangles.
<i>plot</i>	Plot the face.
<i>detach</i>	Detach the face from its body.

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>vertices</i>	List of all vertices of the face.
<i>loops</i>	List of all loops of the face.
<i>color</i>	Get the current color of the face.
<i>opacity</i>	Get the opacity of the face.
<i>bounding_box</i>	Get the bounding box for the face.

Import detail

```
from ansys.geometry.core.designer.face import Face
```

Property detail

property Face.id: str
Face ID.

property `Face.is_reversed`: `bool`

Flag indicating if the face is reversed.

property `Face.body`: `ansys.geometry.core.designer.body.Body`

Body that the face belongs to.

property `Face.shape`: `ansys.geometry.core.shapes-surfaces-trimmed-surface.TrimmedSurface`

Underlying trimmed surface of the face.

If the face is reversed, its shape is a `ReversedTrimmedSurface` type, which handles the direction of the normal vector to ensure it is always facing outward.

Warning

This method is only available starting on Ansys release 24R2.

property `Face.surface_type`: `SurfaceType`

Surface type of the face.

property `Face.area`: `pint.Quantity`

Calculated area of the face.

property `Face.edges`: `list[ansys.geometry.core.designer.edge.Edge]`

List of all edges of the face.

property `Face.vertices`: `list[ansys.geometry.core.designer.vertex.Vertex]`

List of all vertices of the face.

property `Face.loops`: `list[FaceLoop]`

List of all loops of the face.

property `Face.color`: `str`

Get the current color of the face.

Warning

This method is only available starting on Ansys release 25R2.

property `Face.opacity`: `float`

Get the opacity of the face.

property `Face.bounding_box`: `ansys.geometry.core.math.bbox.BoundingBox`

Get the bounding box for the face.

Warning

This method is only available starting on Ansys release 25R2.

Method detail

`Face.get_bounding_box(tight: bool = False) → ansys.geometry.core.math.bbox.BoundingBox`

Get the tight bounding box for the face.

Warning

This method is only available starting on Ansys release 27R1.

Face.**set_color**(color: *str* | *tuple*[float, float, float]) → None

Set the color of the face.

Warning

This method is only available starting on Ansys release 25R2.

Face.**set_opacity**(opacity: *float*) → None

Set the opacity of the face.

Warning

This method is only available starting on Ansys release 25R2.

Face.**normal**(u: *float* = 0.5, v: *float* = 0.5) → *ansys.geometry.core.math.vector.UnitVector3D*

Get the normal direction to the face at certain UV coordinates.

Parameters

u

[*float*, default: 0.5] First coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

v

[*float*, default: 0.5] Second coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

Returns**UnitVector3D**

UnitVector3D object evaluated at the given U and V coordinates. This UnitVector3D object is perpendicular to the surface at the given UV coordinates.

Notes

To properly use this method, you must handle UV coordinates. Thus, you must know how these relate to the underlying Geometry service. It is an advanced method for Geometry experts only.

Face.**point**(u: *float* = 0.5, v: *float* = 0.5) → *ansys.geometry.core.math.point.Point3D*

Get a point of the face evaluated at certain UV coordinates.

Parameters

u

[*float*, default: 0.5] First coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

v

[*float*, default: 0.5] Second coordinate of the 2D representation of a surface in UV space. The default is 0.5, which is the center of the surface.

Returns**Point3D**

Point3D object evaluated at the given UV coordinates.

Notes

To properly use this method, you must handle UV coordinates. Thus, you must know how these relate to the underlying Geometry service. It is an advanced method for Geometry experts only.

`Face.create_isoparametric_curves(use_u_param: bool, parameter: float) → list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]`

Create isoparametric curves at the given proportional parameter.

Typically, only one curve is created, but if the face has a hole, it is possible that more than one curve is created.

Parameters**use_u_param**

[bool] Whether the parameter is the u coordinate or v coordinate. If True, it is the u coordinate. If False, it is the v coordinate.

parameter

[float] Proportional [0-1] parameter to create the one or more curves at.

Returns

`list[TrimmedCurve]`

list of curves that were created.

`Face.setup_offset_relationship(other_face: Face, set_baselines: bool = False, process_adjacent_faces: bool = False) → bool`

Create an offset relationship between two faces.

Parameters**other_face**

[Face] The face to setup an offset relationship with.

set_baselines

[bool, default: False] Automatically set baseline faces.

process_adjacent_faces

[bool, default: False] Look for relationships of the same offset on adjacent faces.

Returns

`bool`

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

`Face.get_named_selections() → list[ansys.geometry.core.designer.selection.NamedSelection]`

Get named selections associated with the edge.

Returns

`list[NamedSelection]`

List of named selections that include the edge.

Face.**tessellate**(*tess_options*: ansys.geometry.core.misc.options.TessellationOptions | *None* = *None*) → *pyvista.PolyData*

Tessellate the face and return the geometry as triangles.

Parameters

tess_options

[TessellationOptions | *None*, default: *None*] A set of options to determine the tessellation quality.

Returns

PolyData

pyvista.PolyData object holding the face.

Notes

The tessellation options are ONLY used if the face has not been tessellated before. If the face has been tessellated before, the stored tessellation is returned.

Face.**plot**(*screenshot*: *str* | *None* = *None*, *use_frame*: *bool* | *None* = *None*, *use_service_colors*: *bool* | *None* = *None*, ***plotting_options*: *dict* | *None*) → *None*

Plot the face.

Parameters

screenshot

[*str*, default: *None*] Path for saving a screenshot of the image that is being represented.

use_frame

[*bool*, default: *None*] Whether to enable the use of *frame*. The default is *None*, in which case the *ansys.tools.visualization_interface.USE_FRAME* global setting is used.

use_service_colors

[*bool*, default: *None*] Whether to use the colors assigned to the face in the service. The default is *None*, in which case the *ansys.geometry.core.USE_SERVICE_COLORS* global setting is used.

****plotting_options**

[*dict*, default: *None*] Keyword arguments for plotting. For allowable keyword arguments, see the *Plotter.add_mesh* method.

Face.**detach**() → *ansys.geometry.core.designer.body.Body* | *None*

Detach the face from its body.

This method will result in the face being turned into a surface body, and the original body will have the face removed.

Returns

Body | None

Body created by the detach. Returns *None* if the operation failed.

Warning

This method is only available starting on Ansys release 27R1.

SurfaceType

class ansys.geometry.core.designer.face.**SurfaceType**(*args, **kws)

Bases: `enum.Enum`

Provides values for the surface types supported.

Overview

Attributes

<code>SURFACETYPE_UNKNOWN</code>
<code>SURFACETYPE_PLANE</code>
<code>SURFACETYPE_CYLINDER</code>
<code>SURFACETYPE_CONE</code>
<code>SURFACETYPE_TORUS</code>
<code>SURFACETYPE_SPHERE</code>
<code>SURFACETYPE_NURBS</code>
<code>SURFACETYPE_PROCEDURAL</code>

Import detail

```
from ansys.geometry.core.designer.face import SurfaceType
```

Attribute detail

`SurfaceType.SURFACETYPE_UNKNOWN = 0`

`SurfaceType.SURFACETYPE_PLANE = 1`

`SurfaceType.SURFACETYPE_CYLINDER = 2`

`SurfaceType.SURFACETYPE_CONE = 3`

`SurfaceType.SURFACETYPE_TORUS = 4`

`SurfaceType.SURFACETYPE_SPHERE = 5`

`SurfaceType.SURFACETYPE_NURBS = 6`

`SurfaceType.SURFACETYPE_PROCEDURAL = 7`

FaceLoopType

class ansys.geometry.core.designer.face.**FaceLoopType**(*args, **kws)

Bases: `enum.Enum`

Provides values for the face loop types supported.

Overview

Attributes

<code>INNER_LOOP</code>
<code>OUTER_LOOP</code>

Import detail

```
from ansys.geometry.core.designer.face import FaceLoopType
```

Attribute detail

FaceLoopType.INNER_LOOP = 'INNER'

FaceLoopType.OUTER_LOOP = 'OUTER'

Description

Module for managing a face.

The geometry_commands.py module

Summary

Classes

<i>GeometryCommands</i>	Provides geometry commands for PyAnsys Geometry.
-------------------------	--

Enums

<i>ExtrudeType</i>	Provides values for extrusion types.
<i>OffsetMode</i>	Provides values for offset modes during extrusions.
<i>FillPatternType</i>	Provides values for types of fill patterns.
<i>DraftSide</i>	Provides values for draft sides.

GeometryCommands

```
class ansys.geometry.core.designer.geometry_commands.GeometryCommands(grpc_client: an-
                                                                    sys.geometry.core.connection.client.GrpcCl
                                                                    _internal_use: bool =
                                                                    False)
```

Provides geometry commands for PyAnsys Geometry.

Parameters

grpc_client

[GrpcClient] gRPC client to use for the geometry commands.

_internal_use

[bool, optional] Internal flag to prevent direct instantiation by users. This parameter is for internal use only.

Raises

GeometryRuntimeError

If the class is instantiated directly by users instead of through the modeler.

Notes

This class should not be instantiated directly. Use `modeler.geometry_commands` instead.

Overview

Methods

<i>chamfer</i>	Create a chamfer on an edge or adjust the chamfer of a face.
<i>fillet</i>	Create a fillet on an edge or adjust the fillet of a face.
<i>full_fillet</i>	Create a full fillet between a collection of faces.
<i>extrude_faces</i>	Extrude a selection of faces.
<i>extrude_faces_up_to</i>	Extrude a selection of faces up to another object.
<i>extrude_edges</i>	Extrude a selection of edges. Provide either a face or a direction and point.
<i>extrude_edges_up_to</i>	Extrude a selection of edges up to another object.
<i>rename_object</i>	Rename an object.
<i>create_linear_pattern</i>	Create a linear pattern. The pattern can be one or two dimensions.
<i>modify_linear_pattern</i>	Modify a linear pattern. Leave an argument at 0 for it to remain unchanged.
<i>create_circular_pattern</i>	Create a circular pattern. The pattern can be one or two dimensions.
<i>modify_circular_pattern</i>	Modify a circular pattern. Leave an argument at 0 for it to remain unchanged.
<i>create_fill_pattern</i>	Create a fill pattern.
<i>update_fill_pattern</i>	Update a fill pattern.
<i>revolve_faces</i>	Revolve face around an axis.
<i>revolve_faces_up_to</i>	Revolve face around an axis up to a certain object.
<i>revolve_faces_by_helix</i>	Revolve face around an axis in a helix shape.
<i>replace_face</i>	Replace a face with another face.
<i>split_body</i>	Split bodies with a plane, slicers, or faces.
<i>get_round_info</i>	Get info on the rounding of a face.
<i>move_translate</i>	Move a selection by a distance in a direction.
<i>move_rotate</i>	Rotate a selection by an angle about a given axis.
<i>offset_faces_set_radius</i>	Offset faces with a radius.
<i>create_align_condition</i>	Create an align condition between two geometry objects.
<i>create_tangent_condition</i>	Create a tangent condition between two geometry objects.
<i>create_orient_condition</i>	Create an orient condition between two geometry objects.
<i>move_imprint_edges</i>	Move the imprint edges in the specified direction by the specified distance.
<i>offset_edges</i>	Offset the specified edges with the specified distance.
<i>draft_faces</i>	Draft the specified faces in the specified direction by the specified angle.
<i>thicken_faces</i>	Thicken the specified faces by the specified thickness in the specified direction.
<i>offset_faces</i>	Offset the specified faces by the specified distance in the specified direction.
<i>revolve_edges</i>	Revolve edges around an axis.
<i>intersect_curve_and_surface</i>	Find the intersection points of a curve and a surface.

continues on next page

Table 4 – continued from previous page

<code>detach_faces</code>	Detach faces on all the bodies/faces from a list.
---------------------------	---

Import detail

```
from ansys.geometry.core.designer.geometry_commands import GeometryCommands
```

Method detail

GeometryCommands.**chamfer**(*selection*: ansys.geometry.core.designer.edge.Edge | list[ansys.geometry.core.designer.edge.Edge] | ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *distance*: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real) → bool

Create a chamfer on an edge or adjust the chamfer of a face.

Parameters

selection

[Edge | list[Edge] | Face | list[Face]] One or more edges or faces to act on.

distance

[Distance | Quantity | Real] Chamfer distance.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**fillet**(*selection*: ansys.geometry.core.designer.edge.Edge | list[ansys.geometry.core.designer.edge.Edge] | ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *radius*: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real) → bool

Create a fillet on an edge or adjust the fillet of a face.

Parameters

selection

[Edge | list[Edge] | Face | list[Face]] One or more edges or faces to act on.

radius

[Distance | Quantity | Real] Fillet radius.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**full_fillet**(*faces: list[ansys.geometry.core.designer.face.Face]*) → bool

Create a full fillet between a collection of faces.

Parameters

faces

[list[Face]] Faces to round.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**extrude_faces**(*faces: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], distance: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real, direction: ansys.geometry.core.math.vector.UnitVector3D = None, extrude_type: ExtrudeType = ExtrudeType.ADD, offset_mode: OffsetMode = OffsetMode.MOVE_FACES_TOGETHER, pull_symmetric: bool = False, copy: bool = False, force_do_as_extrude: bool = False*) → list[ansys.geometry.core.designer.body.Body]

Extrude a selection of faces.

Parameters

faces

[Face | list[Face]] Faces to extrude.

distance

[Distance | Quantity | Real] Distance to extrude.

direction

[UnitVector3D, default: None] Direction of extrusion. If no direction is provided, it will be inferred.

extrude_type

[ExtrudeType, default: ExtrudeType.ADD] Type of extrusion to be performed.

offset_mode

[OffsetMode, default: OffsetMode.MOVE_FACES_TOGETHER] Mode of how to handle off-set relationships.

pull_symmetric

[bool, default: False] Pull symmetrically on both sides if True.

copy

[bool, default: False] Copy the face and move it instead of extruding the original face if True.

force_do_as_extrude

[bool, default: `False`] Forces to do as an extrusion if True, if False allows extrusion by offset.

Returns

`list[Body]`

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.extrude_faces_up_to(faces: ansys.geometry.core.designer.face.Face |
    list[ansys.geometry.core.designer.face.Face], up_to_selection:
    ansys.geometry.core.designer.face.Face |
    ansys.geometry.core.designer.edge.Edge |
    ansys.geometry.core.designer.body.Body, seed_point:
    ansys.geometry.core.math.point.Point3D, direction:
    ansys.geometry.core.math.vector.UnitVector3D, extrude_type:
    ExtrudeType = ExtrudeType.ADD, offset_mode: OffsetMode =
    OffsetMode.MOVE_FACES_TOGETHER, pull_symmetric: bool =
    False, copy: bool = False, force_do_as_extrude: bool = False) →
    list[ansys.geometry.core.designer.body.Body]
```

Extrude a selection of faces up to another object.

Parameters**faces**

[Face | `list[Face]`] Faces to extrude.

up_to_selection

[Face | Edge | Body] The object to pull the faces up to.

seed_point

[Point3D] Origin to define the extrusion.

direction

[UnitVector3D, default: `None`] Direction of extrusion. If no direction is provided, it will be inferred.

extrude_type

[ExtrudeType, default: `ExtrudeType.ADD`] Type of extrusion to be performed.

offset_mode

[OffsetMode, default: `OffsetMode.MOVE_FACES_TOGETHER`] Mode of how to handle offset relationships.

pull_symmetric

[bool, default: `False`] Pull symmetrically on both sides if True.

copy

[bool, default: `False`] Copy the face and move it instead of extruding the original face if True.

force_do_as_extrude

[bool, default: `False`] Forces to do as an extrusion if True, if False allows extrusion by offset.

Returns`list[Body]`

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.extrude_edges(edges: ansys.geometry.core.designer.edge.Edge |  
    list[ansys.geometry.core.designer.edge.Edge], distance:  
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |  
    ansys.geometry.core.typing.Real, from_face:  
    ansys.geometry.core.designer.face.Face = None, from_point:  
    ansys.geometry.core.math.point.Point3D = None, direction:  
    ansys.geometry.core.math.vector.UnitVector3D = None, extrude_type:  
    ExtrudeType = ExtrudeType.ADD, pull_symmetric: bool = False, copy:  
    bool = False, natural_extension: bool = False) →  
    list[ansys.geometry.core.designer.body.Body]
```

Extrude a selection of edges. Provide either a face or a direction and point.

Parameters**edges**[Edge | `list[Edge]`] Edges to extrude.**distance**

[Distance | Quantity | Real] Distance to extrude.

from_face[Face, default: `None`] Face to pull normal from.**from_point**[Point3D, default: `None`] Point to pull from. Must be used with `direction`.**direction**[UnitVector3D, default: `None`] Direction to pull. Must be used with `from_point`.**extrude_type**[ExtrudeType, default: `ExtrudeType.ADD`] Type of extrusion to be performed.**pull_symmetric**[bool, default: `False`] Pull symmetrically on both sides if True.**copy**[bool, default: `False`] Copy the edge and move it instead of extruding the original edge if True.**natural_extension**[bool, default: `False`] Surfaces will extend in a natural or linear shape after exceeding its original range.**Returns**`list[Body]`

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.extrude_edges_up_to(edges: ansys.geometry.core.designer.edge.Edge |
                                     list[ansys.geometry.core.designer.edge.Edge], up_to_selection:
                                     ansys.geometry.core.designer.face.Face |
                                     ansys.geometry.core.designer.edge.Edge |
                                     ansys.geometry.core.designer.body.Body, seed_point:
                                     ansys.geometry.core.math.point.Point3D, direction:
                                     ansys.geometry.core.math.vector.UnitVector3D, extrude_type:
                                     ExtrudeType = ExtrudeType.ADD) →
                                     list[ansys.geometry.core.designer.body.Body]
```

Extrude a selection of edges up to another object.

Parameters

edges

[Edge | list[Edge]] Edges to extrude.

up_to_selection

[Face, default: None] The object to pull the faces up to.

seed_point

[Point3D] Origin to define the extrusion.

direction

[UnitVector3D, default: None] Direction of extrusion.

extrude_type

[ExtrudeType, default: ExtrudeType.ADD] Type of extrusion to be performed.

Returns

list[Body]

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.rename_object(selection: list[ansys.geometry.core.designer.body.Body] |
                                list[ansys.geometry.core.designer.component.Component], name: str) →
                                bool
```

Rename an object.

Parameters

selection

[list[Body] | list[Component]] Selection of the objects to rename.

name

[str] New name for the object.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.create_linear_pattern(selection: ansys.geometry.core.designer.face.Face |
                                       list[ansys.geometry.core.designer.face.Face], linear_direction:
                                       ansys.geometry.core.designer.edge.Edge |
                                       ansys.geometry.core.designer.face.Face, count_x: int, pitch_x:
                                       ansys.geometry.core.misc.measurements.Distance | pint.Quantity
                                       | ansys.geometry.core.typing.Real, two_dimensional: bool =
                                       False, count_y: int = None, pitch_y:
                                       ansys.geometry.core.misc.measurements.Distance | pint.Quantity
                                       | ansys.geometry.core.typing.Real = None) → bool
```

Create a linear pattern. The pattern can be one or two dimensions.

Parameters**selection**

[Face | list[Face]] Faces to create the pattern out of.

linear_direction

[Edge | Face] Direction of the linear pattern, determined by the direction of an edge or face normal.

count_x

[int] How many times the pattern repeats in the x direction.

pitch_x

[Distance | Quantity | Real] The spacing between each pattern member in the x direction.

two_dimensional

[bool, default: False] If True, create a pattern in the x and y direction.

count_y

[int, default: None] How many times the pattern repeats in the y direction.

pitch_y

[Distance | Quantity | Real, default: None] The spacing between each pattern member in the y direction.

Returns**bool**

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.modify_linear_pattern(selection: ansys.geometry.core.designer.face.Face |
                                       list[ansys.geometry.core.designer.face.Face], count_x: int = 0,
                                       pitch_x: ansys.geometry.core.misc.measurements.Distance |
                                       pint.Quantity | ansys.geometry.core.typing.Real = 0.0, count_y:
                                       int = 0, pitch_y:
                                       ansys.geometry.core.misc.measurements.Distance | pint.Quantity
                                       | ansys.geometry.core.typing.Real = 0.0, new_seed_index: int =
                                       0, old_seed_index: int = 0) → bool
```

Modify a linear pattern. Leave an argument at 0 for it to remain unchanged.

Parameters

selection

[Face | list[Face]] Faces that belong to the pattern.

count_x

[int, default: 0] How many times the pattern repeats in the x direction.

pitch_x

[Distance | Quantity | Real, default: 0.0] The spacing between each pattern member in the x direction.

count_y

[int, default: 0] How many times the pattern repeats in the y direction.

pitch_y

[Distance | Quantity | Real, default: 0.0] The spacing between each pattern member in the y direction.

new_seed_index

[int, default: 0] The new seed index of the member.

old_seed_index

[int, default: 0] The old seed index of the member.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.create_circular_pattern(selection: ansys.geometry.core.designer.face.Face |
                                          list[ansys.geometry.core.designer.face.Face], circular_axis:
                                          ansys.geometry.core.designer.edge.Edge |
                                          ansys.geometry.core.shapes.curves.line.Line, circular_count:
                                          int, circular_angle:
                                          ansys.geometry.core.misc.measurements.Angle | pint.Quantity
                                          | ansys.geometry.core.typing.Real, two_dimensional: bool =
                                          False, linear_count: int = None, linear_pitch:
                                          ansys.geometry.core.misc.measurements.Distance |
                                          pint.Quantity | ansys.geometry.core.typing.Real = None,
                                          radial_direction:
                                          ansys.geometry.core.math.vector.UnitVector3D = None) →
                                          bool
```

Create a circular pattern. The pattern can be one or two dimensions.

Parameters

selection

[Face | list[Face]] Faces to create the pattern out of.

circular_axis

[Edge | Line] The axis of the circular pattern.

circular_count

[[int](#)] How many members are in the circular pattern.

circular_angle

[[Angle](#) | [Quantity](#) | [Real](#)] The angular range of the pattern.

two_dimensional

[[bool](#), default: [False](#)] If True, create a two-dimensional pattern.

linear_count

[[int](#), default: [None](#)] How many times the circular pattern repeats along the radial lines for a two-dimensional pattern.

linear_pitch

[[Distance](#) | [Quantity](#) | [Real](#), default: [None](#)] The spacing along the radial lines for a two-dimensional pattern.

radial_direction

[[UnitVector3D](#), default: [None](#)] The direction from the center out for a two-dimensional pattern.

Returns**[bool](#)**

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.modify_circular_pattern(selection: ansys.geometry.core.designer.face.Face |  
                                         list[ansys.geometry.core.designer.face.Face], circular_count:  
                                         int = 0, linear_count: int = 0, step_angle:  
                                         ansys.geometry.core.misc.measurements.Angle | pint.Quantity  
                                         | ansys.geometry.core.typing.Real = 0.0, step_linear:  
                                         ansys.geometry.core.misc.measurements.Distance |  
                                         pint.Quantity | ansys.geometry.core.typing.Real = 0.0) → bool
```

Modify a circular pattern. Leave an argument at 0 for it to remain unchanged.

Parameters**selection**

[[Face](#) | [list](#)[[Face](#)]] Faces that belong to the pattern.

circular_count

[[int](#), default: 0] How many members are in the circular pattern.

linear_count

[[int](#), default: 0] How many times the circular pattern repeats along the radial lines for a two-dimensional pattern.

step_angle

[[Angle](#) | [Quantity](#) | [Real](#), default: 0.0] Defines the circular angle.

step_linear

[[Distance](#) | [Quantity](#) | [Real](#), default: 0.0] Defines the step, along the radial lines, for a pattern dimension greater than 1.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```

GeometryCommands.create_fill_pattern(selection: ansys.geometry.core.designer.face.Face |
                                     list[ansys.geometry.core.designer.face.Face], linear_direction:
                                     ansys.geometry.core.designer.edge.Edge |
                                     ansys.geometry.core.designer.face.Face, fill_pattern_type:
                                     FillPatternType, margin:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real, x_spacing:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real, y_spacing:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real, row_x_offset:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real = 0, row_y_offset:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real = 0, column_x_offset:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real = 0, column_y_offset:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real = 0) → bool

```

Create a fill pattern.

Parameters**selection**

[Face | list[Face]] Faces to create the pattern out of.

linear_direction

[Edge] Direction of the linear pattern, determined by the direction of an edge.

fill_pattern_type

[FillPatternType] The type of fill pattern.

margin

[Distance | Quantity | Real] Margin defining the border of the fill pattern.

x_spacing

[Distance | Quantity | Real] Spacing between the pattern members in the x direction.

y_spacing

[Distance | Quantity | Real] Spacing between the pattern members in the y direction.

row_x_offset

[Distance | Quantity | Real, default: 0] Offset for the rows in the x direction. Only used with FillPattern.SKEWED.

row_y_offset

[Distance | Quantity | Real, default: 0] Offset for the rows in the y direction. Only used with FillPattern.SKEWED.

column_x_offset

[Distance | Quantity | Real, default: 0] Offset for the columns in the x direction. Only

used with FillPattern.SKEWED.

column_y_offset

[Distance | Quantity | Real, default: 0] Offset for the columns in the y direction. Only used with FillPattern.SKEWED.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**update_fill_pattern**(*selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face]) → bool

Update a fill pattern.

When the face that a fill pattern exists upon changes in size, the fill pattern can be updated to fill the new space.

Parameters

selection

[Face | list[Face]] Face(s) that are part of a fill pattern.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**revolve_faces**(*selection*: ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face], *axis*: ansys.geometry.core.designer.edge.Edge | ansys.geometry.core.shapes.curves.line.Line, *angle*: ansys.geometry.core.misc.measurements.Angle | pint.Quantity | ansys.geometry.core.typing.Real, *extrude_type*: ExtrudeType = ExtrudeType.ADD) → list[ansys.geometry.core.designer.body.Body]

Revolve face around an axis.

Parameters

selection

[Face | list[Face]] Face(s) to revolve.

axis

[Edge | Line] Axis of revolution.

angle

[Angle | Quantity | Real] Angular distance to revolve.

extrude_type

[ExtrudeType, default: ExtrudeType.ADD] Type of extrusion to be performed.

Returns`list[Body]`

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.revolve_faces_up_to(selection: ansys.geometry.core.designer.face.Face |
                                       list[ansys.geometry.core.designer.face.Face], up_to:
                                       ansys.geometry.core.designer.face.Face |
                                       ansys.geometry.core.designer.edge.Edge |
                                       ansys.geometry.core.designer.body.Body, axis:
                                       ansys.geometry.core.shapes.curves.line.Line, direction:
                                       ansys.geometry.core.math.vector.UnitVector3D, extrude_type:
                                       ExtrudeType = ExtrudeType.ADD) →
                                       list[ansys.geometry.core.designer.body.Body]
```

Revolve face around an axis up to a certain object.

Parameters**selection**[Face | `list`[Face]] Face(s) to revolve.**up_to**

[Face | Edge | Body] Object to revolve the face up to.

axis

[Line] Axis of revolution.

direction

[UnitVector3D] Direction of extrusion.

extrude_type[ExtrudeType, default: `ExtrudeType.ADD`] Type of extrusion to be performed.**Returns**`list[Body]`

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.revolve_faces_by_helix(selection: ansys.geometry.core.designer.face.Face |
                                         list[ansys.geometry.core.designer.face.Face], axis:
                                         ansys.geometry.core.shapes.curves.line.Line, direction:
                                         ansys.geometry.core.math.vector.UnitVector3D, height:
                                         ansys.geometry.core.misc.measurements.Distance |
                                         pint.Quantity | ansys.geometry.core.typing.Real, pitch:
                                         ansys.geometry.core.misc.measurements.Distance |
                                         pint.Quantity | ansys.geometry.core.typing.Real, taper_angle:
                                         ansys.geometry.core.misc.measurements.Angle | pint.Quantity |
                                         ansys.geometry.core.typing.Real, right_handed: bool,
                                         both_sides: bool, extrude_type: ExtrudeType =
                                         ExtrudeType.ADD) →
                                         list[ansys.geometry.core.designer.body.Body]
```

Revolve face around an axis in a helix shape.

Parameters**selection**

[Face | list[Face]] Face(s) to revolve.

axis

[Line] Axis of revolution.

direction

[UnitVector3D] Direction of extrusion.

height

[Distance | Quantity | Real,] Height of the helix.

pitch

[Distance | Quantity | Real,] Pitch of the helix.

taper_angle

[Angle | Quantity | Real,] Taper angle of the helix.

right_handed

[bool,] Right-handed helix if True, left-handed if False.

both_sides

[bool,] Create on both sides if True, one side if False.

extrude_type

[ExtrudeType, default: *ExtrudeType.ADD*] Type of extrusion to be performed.

Returns**list[Body]**

Bodies created by the extrusion if any.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.replace_face(target_selection: ansys.geometry.core.designer.face.Face |
                                list[ansys.geometry.core.designer.face.Face], replacement_selection:
                                ansys.geometry.core.designer.face.Face |
                                list[ansys.geometry.core.designer.face.Face]) → bool
```

Replace a face with another face.

Parameters

target_selection

[Union[Face, list[Face]]] The face or faces to replace.

replacement_selection

[Union[Face, list[Face]]] The face or faces to replace with.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.split_body(bodies: list[ansys.geometry.core.designer.body.Body], plane:
                             ansys.geometry.core.math.plane.Plane, slicers:
                             ansys.geometry.core.designer.edge.Edge |
                             list[ansys.geometry.core.designer.edge.Edge] |
                             ansys.geometry.core.designer.face.Face |
                             list[ansys.geometry.core.designer.face.Face], faces:
                             list[ansys.geometry.core.designer.face.Face], extendfaces: bool) → bool
```

Split bodies with a plane, slicers, or faces.

Parameters

bodies

[list[Body]] Bodies to split.

plane

[Plane] Plane to split with

slicers

[Edge | list[Edge] | Face | list[Face]] Slicers to split with.

faces

[list[Face]] Faces to split with.

extendFaces

[bool] Extend faces if split with faces.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**get_round_info**(*face*: ansys.geometry.core.designer.face.Face) → tuple[bool, ansys.geometry.core.typing.Real]

Get info on the rounding of a face.

Parameters

Face

The design face to get round info on.

Returns

tuple[bool, Real]

True if round is aligned with face's U-parameter direction, False otherwise. Radius of the round.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**move_translate**(*selection*: ansys.geometry.core.designer.selection.NamedSelection, *direction*: ansys.geometry.core.math.vector.UnitVector3D, *distance*: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real) → bool

Move a selection by a distance in a direction.

Parameters

selection

[NamedSelection] Named selection to move.

direction

[UnitVector3D] Direction to move in.

distance

[Distance | Quantity | Real] Distance to move. Default units are meters.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**move_rotate**(*selection*: ansys.geometry.core.designer.selection.NamedSelection, *axis*: ansys.geometry.core.shapes.curves.line.Line, *angle*: ansys.geometry.core.misc.measurements.Angle | pint.Quantity | ansys.geometry.core.typing.Real) → dict[str, bool | ansys.geometry.core.typing.Real]

Rotate a selection by an angle about a given axis.

Parameters

selection

[NamedSelection] Named selection to move.

axis

[Line] Direction to move in.

Angle

[Angle | Quantity | Real] Angle to rotate by. Default units are radians.

Returns

dict[str, Union[bool, Real]]

Dictionary containing the useful output from the command result. Keys are success, modified_bodies, modified_faces, modified_edges.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.offset_faces_set_radius(faces: ansys.geometry.core.designer.face.Face |
                                          list[ansys.geometry.core.designer.face.Face], radius:
                                          ansys.geometry.core.misc.measurements.Distance |
                                          pint.Quantity | ansys.geometry.core.typing.Real, copy: bool =
                                          False, offset_mode: OffsetMode =
                                          OffsetMode.IGNORE_RELATIONSHIPS, extrude_type:
                                          ExtrudeType = ExtrudeType.FORCE_INDEPENDENT) →
                                          bool
```

Offset faces with a radius.

Parameters**faces**

[Face | list[Face]] Faces to offset.

radius

[Distance | Quantity | Real] Radius of the offset.

copy

[bool, default: False] Copy the face and move it instead of offsetting the original face if True.

offset_mode

[OffsetMode, default: *OffsetMode.MOVE_FACES_TOGETHER*] Mode of how to handle off-set relationships.

extrude_type

[ExtrudeType, default: *ExtrudeType.FORCE_INDEPENDENT*] Type of extrusion to be performed.

Returns

bool

True when successful, False when failed.

Warning

This method is only available starting on Ansys release 25R2.

```
GeometryCommands.create_align_condition(parent_component:
    ansys.geometry.core.designer.component.Component,
    geometry_a: ansys.geometry.core.designer.body.Body |
    ansys.geometry.core.designer.face.Face |
    ansys.geometry.core.designer.edge.Edge, geometry_b:
    ansys.geometry.core.designer.body.Body |
    ansys.geometry.core.designer.face.Face |
    ansys.geometry.core.designer.edge.Edge) →
    ansys.geometry.core.designer.mating_conditions.AlignCondition
```

Create an align condition between two geometry objects.

This will move the objects to be aligned with each other.

Parameters

parent_component

[Component] The common ancestor component of the two geometry objects.

geometry_a

[Body | Face | Edge] The first geometry object to align to the second.

geometry_b

[Body | Face | Edge] The geometry object to be aligned to.

Returns

AlignCondition

The persistent align condition that was created.

Warning

This method is only available starting on Ansys release 26R1.

```
GeometryCommands.create_tangent_condition(parent_component:
    ansys.geometry.core.designer.component.Component,
    geometry_a: ansys.geometry.core.designer.body.Body |
    ansys.geometry.core.designer.face.Face |
    ansys.geometry.core.designer.edge.Edge, geometry_b:
    ansys.geometry.core.designer.body.Body |
    ansys.geometry.core.designer.face.Face |
    ansys.geometry.core.designer.edge.Edge) → an-
    sys.geometry.core.designer.mating_conditions.TangentCondition
```

Create a tangent condition between two geometry objects.

This aligns the objects so that they are tangent.

Parameters

parent_component

[Component] The common ancestor component of the two geometry objects.

geometry_a

[Body | Face | Edge] The first geometry object to tangent the second.

geometry_b

[Body | Face | Edge] The geometry object to be tangent with.

Returns

TangentCondition

The persistent tangent condition that was created.

Warning

This method is only available starting on Ansys release 26R1.

```
GeometryCommands.create_orient_condition(parent_component:
                                         ansys.geometry.core.designer.component.Component,
                                         geometry_a: ansys.geometry.core.designer.body.Body |
                                         ansys.geometry.core.designer.face.Face |
                                         ansys.geometry.core.designer.edge.Edge, geometry_b:
                                         ansys.geometry.core.designer.body.Body |
                                         ansys.geometry.core.designer.face.Face |
                                         ansys.geometry.core.designer.edge.Edge) → an-
                                         sys.geometry.core.designer.mating_conditions.OrientCondition
```

Create an orient condition between two geometry objects.

This rotates the objects so that they are oriented in the same direction.

Parameters**parent_component**

[Component] The common ancestor component of the two geometry objects.

geometry_a

[Body | Face | Edge] The first geometry object to orient with the second.

geometry_b

[Body | Face | Edge] The geometry object to be oriented with.

Returns**OrientCondition**

The persistent orient condition that was created.

Warning

This method is only available starting on Ansys release 26R1.

```
GeometryCommands.move_imprint_edges(edges: list[ansys.geometry.core.designer.edge.Edge], direction:
                                     ansys.geometry.core.math.vector.UnitVector3D, distance:
                                     ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
                                     ansys.geometry.core.typing.Real) → bool
```

Move the imprint edges in the specified direction by the specified distance.

Parameters**edges**

[list[Edge]] The edges to move.

direction

[UnitVector3D] The direction to move the edges.

distance

[Distance | Quantity | Real] The distance to move the edges.

Returns**bool**

Returns True if the edges were moved successfully, False otherwise.

GeometryCommands.**offset_edges**(*edges*: list[ansys.geometry.core.designer.edge.Edge], *offset*: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real) → bool

Offset the specified edges with the specified distance.

Parameters**edges**

[list[Edge]] The edges to offset.

offset

[Distance | Quantity | Real] The distance to offset the edges.

Returns**bool**

Returns True if the edges were offset successfully, False otherwise.

GeometryCommands.**draft_faces**(*faces*: list[ansys.geometry.core.designer.face.Face], *reference_faces*: list[ansys.geometry.core.designer.face.Face], *draft_side*: DraftSide, *angle*: ansys.geometry.core.misc.measurements.Angle | pint.Quantity | ansys.geometry.core.typing.Real, *extrude_type*: ExtrudeType) → list[ansys.geometry.core.designer.face.Face]

Draft the specified faces in the specified direction by the specified angle.

Parameters**faces**

[list[Face]] The faces to draft.

reference_faces

[list[Face]] The reference faces to use for the draft.

draft_side

[DraftSide] The side to draft.

angle

[Angle | Quantity | Real] The angle to draft the faces.

extrude_type

[ExtrudeType] The type of extrusion to use.

Returns**list[Face]**

The faces created by the draft operation.

GeometryCommands.**thicken_faces**(*faces*: list[ansys.geometry.core.designer.face.Face], *direction*: ansys.geometry.core.math.vector.UnitVector3D, *thickness*: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real, *extrude_type*: ExtrudeType, *pull_symmetric*: bool, *select_direction*: bool) → bool

Thicken the specified faces by the specified thickness in the specified direction.

Parameters**faces**

[list[Face]] The faces to thicken.

direction

[UnitVector3D] The direction to thicken the faces.

thickness

[Distance | Quantity | Real] The thickness to apply to the faces.

extrude_type

[ExtrudeType] The type of extrusion to use.

pull_symmetric

[bool] Whether to pull the faces symmetrically.

select_direction

[bool] Whether to select the direction.

Returns**bool**

Returns True if the faces were thickened successfully, False otherwise.

```
GeometryCommands.offset_faces(faces: list[ansys.geometry.core.designer.face.Face], distance:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, direction:
    ansys.geometry.core.math.vector.UnitVector3D, extrude_type: ExtrudeType)
    → None
```

Offset the specified faces by the specified distance in the specified direction.

Parameters**faces**

[list[Face]] The faces to offset.

distance

[Distance | Quantity | Real] The distance to offset the faces.

direction

[UnitVector3D] The direction to offset the faces.

extrude_type

[ExtrudeType] The type of extrusion to use.

Warning

This method is only available starting on Ansys release 26R1.

```
GeometryCommands.revolve_edges(edges: ansys.geometry.core.designer.edge.Edge |
    list[ansys.geometry.core.designer.edge.Edge], axis:
    ansys.geometry.core.shapes.curves.line.Line, angle:
    ansys.geometry.core.misc.measurements.Angle | pint.Quantity |
    ansys.geometry.core.typing.Real, symmetric: bool) → None
```

Revolve edges around an axis.

Parameters**edges**

[Edge | list[Edge]] Edge(s) to revolve.

axis

[Line] Axis of revolution.

angle

[Angle | Quantity | Real] Angular distance to revolve.

symmetric

[bool] Revolve symmetrically if True, one side if False.

Warning

This method is only available starting on Ansys release 25R2.

GeometryCommands.**intersect_curve_and_surface**(*curve*: ansys.geometry.core.shapes.curves.curve.Curve, *surface*: ansys.geometry.core.shapes-surfaces.surface.Surface) → list[ansys.geometry.core.math.point.Point3D]

Find the intersection points of a curve and a surface.

Parameters

curve

[Curve] Curve to intersect.

surface

[Surface] Surface to intersect.

Returns

list[Point3D]

Points of intersection.

Warning

This method is only available starting on Ansys release 26R1. Nurbs curves and surfaces are not supported until Ansys release 27R1.

GeometryCommands.**detach_faces**(*selection*: ansys.geometry.core.designer.body.Body | list[ansys.geometry.core.designer.body.Body] | ansys.geometry.core.designer.face.Face | list[ansys.geometry.core.designer.face.Face]) → list[ansys.geometry.core.designer.body.Body]

Detach faces on all the bodies/faces from a list.

This method will result in a list of new surface bodies:

- If the input is a body, all faces on the body will be detached and the original body will be modified to have only the non-detached faces. The detached faces will be returned as new bodies.
- **If the input is a list of faces:**
 - All the connected faces from the same body will be detached together to form a body.
 - If some of the faces in the list are not connected or are not from the same body, they will be detached separately to form separate bodies.
 - The original body will be modified to have only the non-detached faces. The detached faces will be returned as new bodies.

Parameters

selection

[Body | `list`[Body] | Face | `list`[Face]] Bodies or faces from which we want to detach faces.

Returns

`list`[Body]

Bodies created by the detach if any.

Warning

This method is only available starting on Ansys release 27R1.

ExtrudeType

class ansys.geometry.core.designer.geometry_commands.**ExtrudeType**(*args, **kws)

Bases: `enum.Enum`

Provides values for extrusion types.

Overview**Attributes**

<i>NONE</i>
<i>ADD</i>
<i>CUT</i>
<i>FORCE_ADD</i>
<i>FORCE_CUT</i>
<i>FORCE_INDEPENDENT</i>
<i>FORCE_NEW_SURFACE</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import ExtrudeType
```

Attribute detail

ExtrudeType.**NONE** = 0

ExtrudeType.**ADD** = 1

ExtrudeType.**CUT** = 2

ExtrudeType.**FORCE_ADD** = 3

ExtrudeType.**FORCE_CUT** = 4

ExtrudeType.**FORCE_INDEPENDENT** = 5

ExtrudeType.**FORCE_NEW_SURFACE** = 6

OffsetMode

class ansys.geometry.core.designer.geometry_commands.**OffsetMode**(*args, **kws)

Bases: `enum.Enum`

Provides values for offset modes during extrusions.

Overview

Attributes

<i>IGNORE_RELATIONSHIPS</i>
<i>MOVE_FACES_TOGETHER</i>
<i>MOVE_FACES_APART</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import OffsetMode
```

Attribute detail

`OffsetMode.IGNORE_RELATIONSHIPS = 0`

`OffsetMode.MOVE_FACES_TOGETHER = 1`

`OffsetMode.MOVE_FACES_APART = 2`

FillPatternType

class ansys.geometry.core.designer.geometry_commands.**FillPatternType**(*args, **kws)

Bases: `enum.Enum`

Provides values for types of fill patterns.

Overview

Attributes

<i>GRID</i>
<i>OFFSET</i>
<i>SKEWED</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import FillPatternType
```

Attribute detail

`FillPatternType.GRID = 0`

`FillPatternType.OFFSET = 1`

FillPatternType.SKEWED = 2

DraftSide

class ansys.geometry.core.designer.geometry_commands.DraftSide(*args, **kws)

Bases: `enum.Enum`

Provides values for draft sides.

Overview

Attributes

<i>NO_SPLIT</i>
<i>THIS</i>
<i>OTHER</i>
<i>BACK</i>

Import detail

```
from ansys.geometry.core.designer.geometry_commands import DraftSide
```

Attribute detail

DraftSide.NO_SPLIT = 0

DraftSide.THIS = 1

DraftSide.OTHER = 2

DraftSide.BACK = 3

Description

Provides tools for pulling geometry.

The mating_conditions.py module

Summary

Classes

<i>MatingCondition</i>	MatingCondition dataclass.
<i>AlignCondition</i>	AlignCondition dataclass.
<i>TangentCondition</i>	TangentCondition dataclass.
<i>OrientCondition</i>	OrientCondition dataclass.

MatingCondition

class ansys.geometry.core.designer.mating_conditions.MatingCondition

MatingCondition dataclass.

Overview

Attributes

<i>id</i>
<i>is_deleted</i>
<i>is_enabled</i>
<i>is_satisfied</i>

Import detail

```
from ansys.geometry.core.designer.mating_conditions import MatingCondition
```

Attribute detail

MatingCondition.id: `str`

MatingCondition.is_deleted: `bool`

MatingCondition.is_enabled: `bool`

MatingCondition.is_satisfied: `bool`

AlignCondition

class ansys.geometry.core.designer.mating_conditions.AlignCondition

Bases: MatingCondition

AlignCondition dataclass.

Overview

Attributes

<i>offset</i>
<i>is_reversed</i>
<i>is_valid</i>

Import detail

```
from ansys.geometry.core.designer.mating_conditions import AlignCondition
```

Attribute detail

AlignCondition.offset: `float`

AlignCondition.is_reversed: `bool`

AlignCondition.is_valid: `bool`

TangentCondition

class ansys.geometry.core.designer.mating_conditions.TangentCondition

Bases: MatingCondition

TangentCondition dataclass.

Overview

Attributes

<i>offset</i>
<i>is_reversed</i>
<i>is_valid</i>

Import detail

```
from ansys.geometry.core.designer.mating_conditions import TangentCondition
```

Attribute detail

TangentCondition.offset: float

TangentCondition.is_reversed: bool

TangentCondition.is_valid: bool

OrientCondition

class ansys.geometry.core.designer.mating_conditions.OrientCondition

Bases: MatingCondition

OrientCondition dataclass.

Overview

Attributes

<i>offset</i>
<i>is_reversed</i>
<i>is_valid</i>

Import detail

```
from ansys.geometry.core.designer.mating_conditions import OrientCondition
```

Attribute detail

OrientCondition.offset: float

OrientCondition.is_reversed: bool

`OrientCondition.is_valid: bool`

Description

Provides dataclasses for mating conditions.

The `part.py` module

Summary

Classes

<i>Part</i>	Represents a part master.
<i>MasterComponent</i>	Represents a part occurrence.

Part

```
class ansys.geometry.core.designer.part.Part(id: str, name: str, components: list[MasterComponent],  
                                             bodies:  
                                             list[ansys.geometry.core.designer.body.MasterBody])
```

Represents a part master.

This class should not be accessed by users. The `Part` class holds fundamental data of an assembly.

Parameters

`id`

[`str`] Unique identifier for the part.

`name`

[`str`] Name of the part.

`components`

[`list`[`MasterComponent`]] list of `MasterComponent` children that the part contains.

`bodies`

[`list`[`MasterBody`]] list of `MasterBody` children that the part contains. These are master bodies.

Overview

Properties

<i>id</i>	ID of the part.
<i>name</i>	Name of the part.
<i>components</i>	<code>MasterComponent</code> children that the part contains.
<i>bodies</i>	<code>MasterBody</code> children that the part contains.

Special methods

<code>__repr__</code>	Represent the part as a string.
-----------------------	---------------------------------

Import detail

```
from ansys.geometry.core.designer.part import Part
```

Property detail

property Part.id: `str`

ID of the part.

property Part.name: `str`

Name of the part.

property Part.components: `list[MasterComponent]`

MasterComponent children that the part contains.

property Part.bodies: `list[ansys.geometry.core.designer.body.MasterBody]`

MasterBody children that the part contains.

These are master bodies.

Method detail

Part.__repr__() → `str`

Represent the part as a string.

MasterComponent

```
class ansys.geometry.core.designer.part.MasterComponent(id: str, name: str, part: Part, transform:  
ansys.geometry.core.math.matrix.Matrix44  
= IDENTITY_MATRIX44)
```

Represents a part occurrence.

Parameters

id

[`str`] Unique identifier for the transformed part.

name

[`str`] Name of the transformed part.

part

[`Part`] Reference to the transformed part's master part.

transform

[`Matrix44`] 4x4 transformation matrix from the master part.

Notes

This class should not be accessed by users. It holds the fundamental data of an assembly. Master components wrap parts by adding a transform matrix.

Overview

Properties

<i>id</i>	ID of the transformed part.
<i>name</i>	Name of the transformed part.
<i>occurrences</i>	List of all occurrences of the component.
<i>part</i>	Master part of the transformed part.
<i>transform</i>	4x4 transformation matrix from the master part.

Special methods

<code>__repr__</code>	Represent the master component as a string.
-----------------------	---

Import detail

```
from ansys.geometry.core.designer.part import MasterComponent
```

Property detail

property `MasterComponent.id: str`

ID of the transformed part.

property `MasterComponent.name: str`

Name of the transformed part.

property `MasterComponent.occurrences:`

`list[ansys.geometry.core.designer.component.Component]`

List of all occurrences of the component.

property `MasterComponent.part: Part`

Master part of the transformed part.

property `MasterComponent.transform: ansys.geometry.core.math.matrix.Matrix44`

4x4 transformation matrix from the master part.

Method detail

`MasterComponent.__repr__() → str`

Represent the master component as a string.

Description

Module providing fundamental data of an assembly.

The selection.py module

Summary

Classes

<i>NamedSelection</i>	Represents a single named selection within the design assembly.
-----------------------	---

NamedSelection

```
class ansys.geometry.core.designer.selection.NamedSelection(name: str, design: an-
                                                         sys.geometry.core.designer.design.Design,
                                                         grpc_client: an-
                                                         sys.geometry.core.connection.client.GrpcClient,
                                                         bodies:
                                                         list[ansys.geometry.core.designer.body.Body]
                                                         | None = None, faces:
                                                         list[ansys.geometry.core.designer.face.Face]
                                                         | None = None, edges:
                                                         list[ansys.geometry.core.designer.edge.Edge]
                                                         | None = None, beams:
                                                         list[ansys.geometry.core.designer.beam.Beam]
                                                         | None = None, design_points:
                                                         list[ansys.geometry.core.designer.designpoint.DesignPoi
                                                         | None = None, components:
                                                         list[ansys.geometry.core.designer.component.Component
                                                         | None = None, vertices:
                                                         list[ansys.geometry.core.designer.vertex.Vertex]
                                                         | None = None, preexisting_id: str |
                                                         None = None)
```

Represents a single named selection within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

A named selection organizes one or more design entities together for common actions against the entire group.

Parameters

name

[str] User-defined name for the named selection.

design

[Design] Design instance to which the named selection belongs.

grpc_client

[GrpcClient] Active supporting Geometry service instance for design modeling.

bodies

[list[Body], default: None] All bodies to include in the named selection.

faces

[list[Face], default: None] All faces to include in the named selection.

edges

[list[Edge], default: None] All edges to include in the named selection.

beams

[list[Beam], default: None] All beams to include in the named selection.

design_points

[list[DesignPoints], default: None] All design points to include in the named selection.

components: list[Component], default: None

All components to include in the named selection.

vertices: list[Vertex], default: None

All vertices to include in the named selection.

Overview

Methods

<code>add_members</code>	Add members to the named selection.
<code>remove_members</code>	Remove members from the named selection.

Properties

<code>id</code>	ID of the named selection.
<code>name</code>	Name of the named selection.
<code>bodies</code>	All bodies in the named selection.
<code>faces</code>	All faces in the named selection.
<code>edges</code>	All edges in the named selection.
<code>beams</code>	All beams in the named selection.
<code>design_points</code>	All design points in the named selection.
<code>components</code>	All components in the named selection.
<code>vertices</code>	All vertices in the named selection.

Special methods

<code>__repr__</code>	Represent the NamedSelection as a string.
-----------------------	---

Import detail

```
from ansys.geometry.core.designer.selection import NamedSelection
```

Property detail

property `NamedSelection.id`: `str`

ID of the named selection.

property `NamedSelection.name`: `str`

Name of the named selection.

property `NamedSelection.bodies`: `list[ansys.geometry.core.designer.body.Body]`

All bodies in the named selection.

property `NamedSelection.faces`: `list[ansys.geometry.core.designer.face.Face]`

All faces in the named selection.

property `NamedSelection.edges`: `list[ansys.geometry.core.designer.edge.Edge]`

All edges in the named selection.

property `NamedSelection.beams`: `list[ansys.geometry.core.designer.beam.Beam]`

All beams in the named selection.

property `NamedSelection.design_points`:

`list[ansys.geometry.core.designer.designpoint.DesignPoint]`

All design points in the named selection.

property NamedSelection.components:
`list[ansys.geometry.core.designer.component.Component]`

All components in the named selection.

property NamedSelection.vertices: `list[ansys.geometry.core.designer.vertex.Vertex]`

All vertices in the named selection.

Method detail

NamedSelection.add_members(*bodies: list[ansys.geometry.core.designer.body.Body] | None = None, faces: list[ansys.geometry.core.designer.face.Face] | None = None, edges: list[ansys.geometry.core.designer.edge.Edge] | None = None, beams: list[ansys.geometry.core.designer.beam.Beam] | None = None, design_points: list[ansys.geometry.core.designer.designpoint.DesignPoint] | None = None, components: list[ansys.geometry.core.designer.component.Component] | None = None, vertices: list[ansys.geometry.core.designer.vertex.Vertex] | None = None*)
 → NamedSelection

Add members to the named selection.

Parameters

bodies

`[list[Body], default: None]`

All bodies to add to the named selection.

faces

`[list[Face], default: None]`

All faces to add to the named selection.

edges

`[list[Edge], default: None]`

All edges to add to the named selection.

beams

`[list[Beam], default: None]`

All beams to add to the named selection.

design_points

`[list[DesignPoints], default: None]`

All design points to add to the named selection.

components: `list[Component]`, default: None

All components to add to the named selection.

vertices: `list[Vertex]`, default: None

All vertices to add to the named selection.

Returns

NamedSelection

The new named selection with the added members. Changes are also reflected on the same named selection upon which the method is called.

Notes

Named selections are immutable. This method creates a new named selection with the added members.

```
NamedSelection.remove_members(members: list[ansys.geometry.core.designer.body.Body |
ansys.geometry.core.designer.face.Face |
ansys.geometry.core.designer.edge.Edge |
ansys.geometry.core.designer.beam.Beam |
ansys.geometry.core.designer.designpoint.DesignPoint |
ansys.geometry.core.designer.component.Component |
ansys.geometry.core.designer.vertex.Vertex]) → NamedSelection
```

Remove members from the named selection.

Parameters

members

[list of Body, Face, Edge, Beam, DesignPoint, Component, or Vertex] The members to remove from the named selection.

Returns

NamedSelection

The new named selection with the members removed. Changes are also reflected on the same named selection upon which the method is called.

```
NamedSelection.__repr__() → str
```

Represent the NamedSelection as a string.

Description

Module for creating a named selection.

The vertex.py module

Summary

Classes

<i>Vertex</i>	Represents a single vertex of a body within the design assembly.
---------------	--

Vertex

```
class ansys.geometry.core.designer.vertex.Vertex(id: str, position: numpy.ndarray |
ansys.geometry.core.typing.RealSequence, body:
ansys.geometry.core.designer.body.Body)
```

Bases: *ansys.geometry.core.math.point.Point3D*

Represents a single vertex of a body within the design assembly.

This class synchronizes to a design within a supporting Geometry service instance.

Parameters

id

[str] The unique identifier for the vertex.

position

[np.ndarray or RealSequence] The position of the vertex in 3D space.

Overview

Methods

<code>get_named_selections</code>	Get all named selections that include this vertex.
-----------------------------------	--

Properties

<code>id</code>	Get the unique identifier of the vertex.
<code>body</code>	Get the body this vertex belongs to.

Special methods

<code>__repr__</code>	Return a string representation of the vertex.
-----------------------	---

Import detail

```
from ansys.geometry.core.designer.vertex import Vertex
```

Property detail

property `Vertex.id: str`

Get the unique identifier of the vertex.

property `Vertex.body: ansys.geometry.core.designer.body.Body`

Get the body this vertex belongs to.

Method detail

`Vertex.get_named_selections()` → `list[ansys.geometry.core.designer.selection.NamedSelection]`

Get all named selections that include this vertex.

Returns

`list["NamedSelection"]`

List of named selections that include this vertex.

`Vertex.__repr__()` → `str`

Return a string representation of the vertex.

Description

Module for managing a vertex.

Description

PyAnsys Geometry designer subpackage.

The materials package

Summary

Submodules

<i>material</i>	Provides the data structure for material and material properties.
<i>property</i>	Provides the MaterialProperty class.

The material.py module

Summary

Classes

<i>Material</i>	Provides the data structure for a material.
-----------------	---

Material

class ansys.geometry.core.materials.material.**Material**(*name: str, density: pint.Quantity, additional_properties: collections.abc.Sequence[ansys.geometry.core.materials.property.MaterialProperty] | None = None*)

Provides the data structure for a material.

Parameters

name: str

Material name.

density: ~pint.Quantity

Material density.

additional_properties: Sequence[MaterialProperty], default: None

Additional material properties.

Overview

Methods

<i>add_property</i>	Add a material property to the Material class.
---------------------	--

Properties

<i>properties</i>	Dictionary of the material property type and material properties.
<i>name</i>	Material name.

Import detail

```
from ansys.geometry.core.materials.material import Material
```

Property detail

property `Material.properties:`
`dict[ansys.geometry.core.materials.property.MaterialPropertyType,`
`ansys.geometry.core.materials.property.MaterialProperty]`

Dictionary of the material property type and material properties.

property `Material.name: str`

Material name.

Method detail

`Material.add_property(type: ansys.geometry.core.materials.property.MaterialPropertyType, name: str,`
`quantity: pint.Quantity) → None`

Add a material property to the Material class.

Parameters

type

[MaterialPropertyType] Material property type.

name: str

Material name.

quantity: ~pint.Quantity

Material value and unit.

Description

Provides the data structure for material and material properties.

The property.py module

Summary

Classes

<i>MaterialProperty</i>	Provides the data structure for a material property.
-------------------------	--

Enums

<i>MaterialPropertyType</i>	Enum holding the possible values for MaterialProperty objects.
-----------------------------	--

MaterialProperty

class `ansys.geometry.core.materials.property.MaterialProperty`(*type: MaterialPropertyType | str,*
name: str, quantity: pint.Quantity |
ansys.geometry.core.typing.Real)

Provides the data structure for a material property.

Parameters

type

[MaterialPropertyType | str] Type of the material property. If the type is a string, it

must be a valid material property type - though it might not be supported by the Material-PropertyType enum.

name: `str`

Material property name.

quantity: `~pint.Quantity | Real`

Value and unit in case of a supported Quantity. If the type is not supported, it must be a Real value (float or integer).

Overview

Properties

<code>type</code>	Material property ID.
<code>name</code>	Material property name.
<code>quantity</code>	Material property quantity and unit.

Import detail

```
from ansys.geometry.core.materials.property import MaterialProperty
```

Property detail

property `MaterialProperty.type:` `MaterialPropertyType | str`

Material property ID.

If the type is not supported, it will be a string.

property `MaterialProperty.name:` `str`

Material property name.

property `MaterialProperty.quantity:` `pint.Quantity | ansys.geometry.core.typing.Real`

Material property quantity and unit.

If the type is not supported, it will be a Real value (float or integer).

MaterialPropertyType

class `ansys.geometry.core.materials.property.MaterialPropertyType(*args, **kwargs)`

Bases: `enum.Enum`

Enum holding the possible values for MaterialProperty objects.

Overview

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>

Static methods

<code>from_id</code>	Return the MaterialPropertyType value from the service.
----------------------	---

Import detail

```
from ansys.geometry.core.materials.property import MaterialPropertyType
```

Attribute detail

`MaterialPropertyType.DENSITY = 'Density'`

`MaterialPropertyType.ELASTIC_MODULUS = 'ElasticModulus'`

`MaterialPropertyType.POISSON_RATIO = 'PoissonsRatio'`

`MaterialPropertyType.SHEAR_MODULUS = 'ShearModulus'`

`MaterialPropertyType.SPECIFIC_HEAT = 'SpecificHeat'`

`MaterialPropertyType.TENSILE_STRENGTH = 'TensileStrength'`

`MaterialPropertyType.THERMAL_CONDUCTIVITY = 'ThermalConductivity'`

Method detail

static `MaterialPropertyType.from_id(id: str) → MaterialPropertyType`

Return the MaterialPropertyType value from the service.

Parameters

id

[str] Geometry Service string representation of a property type.

Returns

MaterialPropertyType

Common name for property type.

Description

Provides the MaterialProperty class.

Description

PyAnsys Geometry materials subpackage.

The math package

Summary

Submodules

<i>bbox</i>	Provides for managing a bounding box.
<i>constants</i>	Provides mathematical constants.
<i>frame</i>	Provides for managing a frame.
<i>matrix</i>	Provides matrix primitive representations.
<i>misc</i>	Provides auxiliary math functions for PyAnsys Geometry.
<i>plane</i>	Provides primitive representation of a 2D plane in 3D space.
<i>point</i>	Provides geometry primitive representation for 2D and 3D points.
<i>vector</i>	Provides for creating and managing 2D and 3D vectors.

The `bbox.py` module

Summary

Classes

<i>BoundingBox</i>	Maintains the box structure for Bounding Boxes.
--------------------	---

BoundingBox

```
class ansys.geometry.core.math.bbox.BoundingBox(min_corner:
                                                ansys.geometry.core.math.point.Point3D,
                                                max_corner:
                                                ansys.geometry.core.math.point.Point3D, center:
                                                ansys.geometry.core.math.point.Point3D = None)
```

Maintains the box structure for Bounding Boxes.

Parameters

min_corner
[Point3D] Minimum corner for the box.

max_corner
[Point3D] Maximum corner for the box.

center
[Point3D] Center of the box.

Overview

Methods

<i>contains_point</i>	Evaluate whether a point lies within the box.
<i>contains_point_components</i>	Check if point components are within box.

Properties

<i>min_corner</i>	Minimum corner of the bounding box.
<i>max_corner</i>	Maximum corner of the bounding box.
<i>center</i>	Center of the bounding box.

Static methods

<code>intersect_bboxes</code>	Find the intersection of 2 BoundingBox objects.
-------------------------------	---

Special methods

<code>__eq__</code>	Equals operator for the BoundingBox class.
<code>__ne__</code>	Not equals operator for the BoundingBox class.

Import detail

```
from ansys.geometry.core.math.bbox import BoundingBox
```

Property detail

property BoundingBox.**min_corner**: *ansys.geometry.core.math.point.Point3D*
Minimum corner of the bounding box.

property BoundingBox.**max_corner**: *ansys.geometry.core.math.point.Point3D*
Maximum corner of the bounding box.

property BoundingBox.**center**: *ansys.geometry.core.math.point.Point3D*
Center of the bounding box.

Method detail

BoundingBox.**contains_point**(*point*: *ansys.geometry.core.math.point.Point3D*) → bool
Evaluate whether a point lies within the box.

Parameters

point
[Point3D] Point to compare against the bounds.

Returns

bool
True if the point is contained in the bounding box. Otherwise, False.

BoundingBox.**contains_point_components**(*x*: *ansys.geometry.core.typing.Real*, *y*: *ansys.geometry.core.typing.Real*, *z*: *ansys.geometry.core.typing.Real*) → bool

Check if point components are within box.

Parameters

x
[Real] Point X component to compare against the bounds.

y
[Real] Point Y component to compare against the bounds.

z
[Real] Point Z component to compare against the bounds.

Returns

bool

True if the components are contained in the bounding box. Otherwise, False.

BoundingBox.__eq__(*other*: [BoundingBox](#)) → bool

Equals operator for the BoundingBox class.

BoundingBox.__ne__(*other*: [BoundingBox](#)) → bool

Not equals operator for the BoundingBox class.

static BoundingBox.intersect_bboxes(*box_1*: [BoundingBox](#), *box_2*: [BoundingBox](#)) → None | [BoundingBox](#)

Find the intersection of 2 BoundingBox objects.

Parameters**box_1: BoundingBox**

The box to consider the intersection of with respect to box_2.

box_2: BoundingBox

The box to consider the intersection of with respect to box_1.

Returns**BoundingBox:**

The box representing the intersection of the two passed in boxes.

Description

Provides for managing a bounding box.

The constants.py module**Summary****Constants**

DEFAULT_POINT3D	Default value for a 3D point.
DEFAULT_POINT2D	Default value for a 2D point.
IDENTITY_MATRIX33	Identity for a Matrix33 object.
IDENTITY_MATRIX44	Identity for a Matrix44 object.
UNITVECTOR3D_X	Default 3D unit vector in the Cartesian traditional X direction.
UNITVECTOR3D_Y	Default 3D unit vector in the Cartesian traditional Y direction.
UNITVECTOR3D_Z	Default 3D unit vector in the Cartesian traditional Z direction.
UNITVECTOR2D_X	Default 2D unit vector in the Cartesian traditional X direction.
UNITVECTOR2D_Y	Default 2D unit vector in the Cartesian traditional Y direction.
ZERO_VECTOR3D	Zero-valued Vector3D object.
ZERO_VECTOR2D	Zero-valued Vector2D object.
ZERO_POINT3D	Zero-valued Point3D object.
ZERO_POINT2D	Zero-valued Point2D object.

Description

Provides mathematical constants.

Module detail

`ansys.geometry.core.math.constants.DEFAULT_POINT3D`

Default value for a 3D point.

`ansys.geometry.core.math.constants.DEFAULT_POINT2D`

Default value for a 2D point.

`ansys.geometry.core.math.constants.IDENTITY_MATRIX33`

Identity for a Matrix33 object.

`ansys.geometry.core.math.constants.IDENTITY_MATRIX44`

Identity for a Matrix44 object.

`ansys.geometry.core.math.constants.UNITVECTOR3D_X`

Default 3D unit vector in the Cartesian traditional X direction.

`ansys.geometry.core.math.constants.UNITVECTOR3D_Y`

Default 3D unit vector in the Cartesian traditional Y direction.

`ansys.geometry.core.math.constants.UNITVECTOR3D_Z`

Default 3D unit vector in the Cartesian traditional Z direction.

`ansys.geometry.core.math.constants.UNITVECTOR2D_X`

Default 2D unit vector in the Cartesian traditional X direction.

`ansys.geometry.core.math.constants.UNITVECTOR2D_Y`

Default 2D unit vector in the Cartesian traditional Y direction.

`ansys.geometry.core.math.constants.ZERO_VECTOR3D`

Zero-valued Vector3D object.

`ansys.geometry.core.math.constants.ZERO_VECTOR2D`

Zero-valued Vector2D object.

`ansys.geometry.core.math.constants.ZERO_POINT3D`

Zero-valued Point3D object.

`ansys.geometry.core.math.constants.ZERO_POINT2D`

Zero-valued Point2D object.

The `frame.py` module

Summary

Classes

<i>Frame</i>	Representation of a frame.
--------------	----------------------------

Frame


```

class ansys.geometry.core.math.frame.Frame(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D =
    ZERO_POINT3D, direction_x: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_X, direction_y: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_Y)

```

Representation of a frame.

Parameters

origin

[[ndarray](#) | [RealSequence](#) | [Point3D](#), default: [ZERO_POINT3D](#)] Centered origin of the frame. The default is [ZERO_POINT3D](#), which is the Cartesian origin.

direction_x

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#), default: [UNITVECTOR3D_X](#)] X-axis direction.

direction_y

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#), default: [UNITVECTOR3D_Y](#)] Y-axis direction.

Overview

Methods

transform_point2d_local_to_global	Transform a 2D point to a global 3D point.
---	--

Properties

origin	Origin of the frame.
direction_x	X-axis direction of the frame.
direction_y	Y-axis direction of the frame.
direction_z	Z-axis direction of the frame.
global_to_local_rotation	Global to local space transformation matrix.
local_to_global_rotation	Local to global space transformation matrix.
transformation_matrix	Full 4x4 transformation matrix.

Special methods

__eq__	Equals operator for the Frame class.
__ne__	Not equals operator for the Frame class.

Import detail

```
from ansys.geometry.core.math.frame import Frame
```

Property detail

property `Frame.origin`: *ansys.geometry.core.math.point.Point3D*

Origin of the frame.

property `Frame.direction_x`: *ansys.geometry.core.math.vector.UnitVector3D*

X-axis direction of the frame.

property `Frame.direction_y`: *ansys.geometry.core.math.vector.UnitVector3D*

Y-axis direction of the frame.

property `Frame.direction_z`: *ansys.geometry.core.math.vector.UnitVector3D*

Z-axis direction of the frame.

property `Frame.global_to_local_rotation`: *ansys.geometry.core.math.matrix.Matrix33*

Global to local space transformation matrix.

Returns

Matrix33

3x3 matrix representing the transformation from global to local coordinate space, excluding origin translation.

property `Frame.local_to_global_rotation`: *ansys.geometry.core.math.matrix.Matrix33*

Local to global space transformation matrix.

Returns

Matrix33

3x3 matrix representing the transformation from local to global coordinate space.

property `Frame.transformation_matrix`: *ansys.geometry.core.math.matrix.Matrix44*

Full 4x4 transformation matrix.

Returns

Matrix44

4x4 matrix representing the transformation from global to local coordinate space.

Method detail

`Frame.transform_point2d_local_to_global`(*point*: *ansys.geometry.core.math.point.Point2D*) → *ansys.geometry.core.math.point.Point3D*

Transform a 2D point to a global 3D point.

This method transforms a local, plane-contained `Point2D` object in the global coordinate system, thus representing it as a `Point3D` object.

Parameters

point

[`Point2D`] `Point2D` local object to express in global coordinates.

Returns

Point3D

Global coordinates for the 3D point.

`Frame.__eq__(other: Frame) → bool`

Equals operator for the Frame class.

`Frame.__ne__(other: Frame) → bool`

Not equals operator for the Frame class.

Description

Provides for managing a frame.

The matrix.py module**Summary****Classes**

<i>Matrix</i>	Provides matrix representation.
<i>Matrix33</i>	Provides 3x3 matrix representation.
<i>Matrix44</i>	Provides 4x4 matrix representation.

Constants

<i>DEFAULT_MATRIX33</i>	Default value of the 3x3 identity matrix for the Matrix33 class.
<i>DEFAULT_MATRIX44</i>	Default value of the 4x4 identity matrix for the Matrix44 class.

Matrix

class ansys.geometry.core.math.matrix.**Matrix**(*shape, dtype=float, buffer=None, offset=0, strides=None, order=None*)

Bases: `numpy.ndarray`

Provides matrix representation.

Parameters**input**

[`ndarray` | `RealSequence`] Matrix arguments as a `np.ndarray` class.

Overview**Methods**

<i>determinant</i>	Get the determinant of the matrix.
<i>inverse</i>	Provide the inverse of the matrix.

Special methods

<code>__mul__</code>	Get the multiplication of the matrix.
<code>__eq__</code>	Equals operator for the <code>Matrix</code> class.
<code>__ne__</code>	Not equals operator for the <code>Matrix</code> class.

Import detail

```
from ansys.geometry.core.math.matrix import Matrix
```

Method detail

`Matrix.determinant()` → *ansys.geometry.core.typing.Real*

Get the determinant of the matrix.

`Matrix.inverse()` → *Matrix*

Provide the inverse of the matrix.

`Matrix.__mul__(other: Matrix | numpy.ndarray)` → *Matrix*

Get the multiplication of the matrix.

`Matrix.__eq__(other: Matrix)` → *bool*

Equals operator for the `Matrix` class.

`Matrix.__ne__(other: Matrix)` → *bool*

Not equals operator for the `Matrix` class.

Matrix33

```
class ansys.geometry.core.math.matrix.Matrix33(shape, dtype=float, buffer=None, offset=0,
                                              strides=None, order=None)
```

Bases: `Matrix`

Provides 3x3 matrix representation.

Parameters

input

[*ndarray* | *RealSequence* | *Matrix*, default: `DEFAULT_MATRIX33`] Matrix arguments as a *np.ndarray* class.

Import detail

```
from ansys.geometry.core.math.matrix import Matrix33
```

Matrix44

```
class ansys.geometry.core.math.matrix.Matrix44(shape, dtype=float, buffer=None, offset=0,
                                              strides=None, order=None)
```

Bases: `Matrix`

Provides 4x4 matrix representation.

Parameters

input

[[ndarray](#) | [RealSequence](#) | [Matrix](#), default: `DEFAULT_MATRIX44`] Matrix arguments as a `np.ndarray` class.

Overview**Constructors**

create_translation	Create a matrix representing the specified translation.
create_rotation	Create a matrix representing the specified rotation.
create_matrix_from_rotation_about_axis	Create a matrix representing a rotation about a given axis.
create_matrix_from_mapping	Create a matrix representing the specified mapping.

Methods

is_translation	Check if the matrix represents a translation.
--------------------------------	---

Import detail

```
from ansys.geometry.core.math.matrix import Matrix44
```

Method detail

classmethod `Matrix44.create_translation(translation: ansys.geometry.core.math.vector.Vector3D) → Matrix44`

Create a matrix representing the specified translation.

Parameters**translation**

[[Vector3D](#)] The translation vector representing the translation. The components of the vector should be in meters.

Returns**Matrix44**

A 4x4 matrix representing the translation.

Examples

```
>>> translation_vector = Vector3D(1.0, 2.0, 3.0)
>>> translation_matrix = Matrix44.create_translation(translation_vector)
>>> print(translation_matrix)
[[1.  0.  0.  1.]
 [0.  1.  0.  2.]
 [0.  0.  1.  3.]
 [0.  0.  0.  1.]]
```

Matrix44.is_translation(including_identity: *bool* = *False*) → *bool*

Check if the matrix represents a translation.

This method checks if the matrix represents a translation transformation. A translation matrix has the following form:

[1 0 0 tx] [0 1 0 ty] [0 0 1 tz] [0 0 0 1]

Parameters

including_identity

[bool, optional] If True, the method will return True for the identity matrix as well. If False, the method will return False for the identity matrix.

Returns

bool

True if the matrix represents a translation, False otherwise.

Examples

```
>>> matrix = Matrix44([[1, 0, 0, 5], [0, 1, 0, 3], [0, 0, 1, 2], [0, 0, 0, 1]])
>>> matrix.is_translation()
True
>>> identity_matrix = Matrix44([[1, 0, 0, 0], [0, 1, 0, 0], [0, 0, 1, 0], [0, 0, 0, 1]])
>>> identity_matrix.is_translation()
False
>>> identity_matrix.is_translation(including_identity=True)
True
```

classmethod `Matrix44.create_rotation`(*direction_x*: `ansys.geometry.core.math.vector.Vector3D`,
direction_y: `ansys.geometry.core.math.vector.Vector3D`, *direction_z*:
`ansys.geometry.core.math.vector.Vector3D = None`) → *Matrix44*

Create a matrix representing the specified rotation.

Parameters

direction_x

[Vector3D] The X direction vector.

direction_y

[Vector3D] The Y direction vector.

direction_z

[Vector3D, optional] The Z direction vector. If not provided, it will be calculated as the cross product of *direction_x* and *direction_y*.

Returns

Matrix44

A 4x4 matrix representing the rotation.

Examples

```
>>> direction_x = Vector3D(1.0, 0.0, 0.0)
>>> direction_y = Vector3D(0.0, 1.0, 0.0)
>>> rotation_matrix = Matrix44.create_rotation(direction_x, direction_y)
>>> print(rotation_matrix)
[[1. 0. 0. 0.]
 [0. 1. 0. 0.]
 [0. 0. 1. 0.]
 [0. 0. 0. 1.]]
```

classmethod `Matrix44.create_matrix_from_rotation_about_axis`(*axis*: `ansys.geometry.core.math.vector.Vector3D`,
angle: `float`) → *Matrix44*

Create a matrix representing a rotation about a given axis.

Parameters

axis

[`Vector3D`] The axis of rotation.

angle

[`float`] The angle of rotation in radians.

Returns

Matrix44

A 4x4 matrix representing the rotation.

classmethod `Matrix44.create_matrix_from_mapping`(*frame*: `ansys.geometry.core.math.frame.Frame`) →
Matrix44

Create a matrix representing the specified mapping.

Parameters

frame

[`Frame`] The frame containing the origin and direction vectors.

Returns

Matrix44

A 4x4 matrix representing the translation and rotation defined by the frame.

Description

Provides matrix primitive representations.

Module detail

`ansys.geometry.core.math.matrix.DEFAULT_MATRIX33`

Default value of the 3x3 identity matrix for the `Matrix33` class.

`ansys.geometry.core.math.matrix.DEFAULT_MATRIX44`

Default value of the 4x4 identity matrix for the `Matrix44` class.

The `misc.py` module

Summary

Functions

<code>get_two_circle_intersections</code>	Get the intersection points of two circles.
<code>intersect_interval</code>	Find the intersection of two intervals.

Description

Provides auxiliary math functions for PyAnsys Geometry.

Module detail

`ansys.geometry.core.math.misc.get_two_circle_intersections`(*x0*: `ansys.geometry.core.typing.Real`,
y0: `ansys.geometry.core.typing.Real`,
r0: `ansys.geometry.core.typing.Real`,
x1: `ansys.geometry.core.typing.Real`,
y1: `ansys.geometry.core.typing.Real`,
r1: `ansys.geometry.core.typing.Real`) → `tuple[tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real], tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real]] | None`

Get the intersection points of two circles.

Parameters

x0
`[Real]` x coordinate of the first circle.

y0
`[Real]` y coordinate of the first circle.

r0
`[Real]` Radius of the first circle.

x1
`[Real]` x coordinate of the second circle.

y1
`[Real]` y coordinate of the second circle.

r1
`[Real]` Radius of the second circle.

Returns

`tuple[tuple[Real, Real], tuple[Real, Real]] | None`
 Intersection points of the two circles if they intersect. The points are returned as ((*x3*, *y3*), (*x4*, *y4*)), where (*x3*, *y3*) and (*x4*, *y4*) are the intersection points of the two circles. If the circles do not intersect, then `None` is returned.

Notes

This function is based on the following StackOverflow post: <https://stackoverflow.com/questions/55816902/finding-the-intersection-of-two-circles>

That post is based on the following implementation: <https://paulbourke.net/geometry/circlesphere/>

`ansys.geometry.core.math.misc.intersect_interval`(*first_min*, *second_min*, *first_max*, *second_max*) → `tuple[bool, ansys.geometry.core.typing.Real, ansys.geometry.core.typing.Real]`

Find the intersection of two intervals.

Parameters

first_min

[Real] The minimum value of the first interval.

second_min

[Real] The minimum value of the second interval.

first_max

[Real] The maximum value of the first interval.

second_max

[Real] The maximum value of the second interval.

Returns

tuple[bool, Real, Real]

Tuple with a boolean to indicate whether the intervals intersect, the minimum value of the intersection interval, and the maximum value of the intersection interval. If they do not intersect, then the boolean is False and the other values are 0.

The plane.py module

Summary

Classes

<i>Plane</i> Provides primitive representation of a 2D plane in 3D space.

Plane

```
class ansys.geometry.core.math.plane.Plane(origin: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.point.Point3D =
ZERO_POINT3D, direction_x: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_X, direction_y: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_Y)
```

Bases: *ansys.geometry.core.math.frame.Frame*

Provides primitive representation of a 2D plane in 3D space.

Parameters

origin

[ndarray | RealSequence | Point3D, default: ZERO_POINT3D] Centered origin of the frame. The default is ZERO_POINT3D, which is the Cartesian origin.

direction_x

[ndarray | RealSequence | UnitVector3D | Vector3D, default: UNITVECTOR3D_X] X-axis direction.

direction_y

[ndarray | RealSequence | UnitVector3D | Vector3D, default: UNITVECTOR3D_Y] Y-axis direction.

Overview

Methods

<code>is_point_contained</code>	Check if a 3D point is contained in the plane.
---------------------------------	--

Properties

<code>normal</code>	Calculate the normal vector of the plane.
---------------------	---

Special methods

<code>__eq__</code>	Equals operator for the 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

<code>Point2D</code>	Provides geometry primitive representation for a 2D point.
<code>Point3D</code>	Provides geometry primitive representation for a 3D point.

Constants

<code>DEFAULT_POINT2D_VALUES</code>	Default values for a 2D point.
<code>DEFAULT_POINT3D_VALUES</code>	Default values for a 3D point.
<code>BASE_UNIT_LENGTH</code>	Default value for the length of the base unit.

Point2D

```
class ansys.geometry.core.math.point.Point2D(input: numpy.ndarray |
                                              ansys.geometry.core.typing.RealSequence =
                                              DEFAULT_POINT2D_VALUES, unit: pint.Unit | None =
                                              None)
```

Bases: `numpy.ndarray`, `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides geometry primitive representation for a 2D point.

Parameters

input

[`ndarray` | `RealSequence`, default: `DEFAULT_POINT2D_VALUES`] Direction arguments, either as a `numpy.ndarray` class or as a `RealSequence`.

unit

[`Unit` | `None`, default: `DEFAULT_UNITS.LENGTH`] Units for defining 2D point values. If not specified, the default unit is `DEFAULT_UNITS.LENGTH`.

Overview

Methods

<code>unit</code>	Get the unit of the object.
<code>base_unit</code>	Get the base unit of the object.

Properties

<code>x</code>	X plane component value.
<code>y</code>	Y plane component value.

Attributes

flat

Special methods

<code>__eq__</code>	Equals operator for the <code>Point2D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Point2D</code> class.
<code>__add__</code>	Add operation for the <code>Point2D</code> class.
<code>__sub__</code>	Subtraction operation for the <code>Point2D</code> class.

Import detail

```
from ansys.geometry.core.math.point import Point2D
```

Property detail

property `Point2D.x`: `pint.Quantity`

X plane component value.

property `Point2D.y`: `pint.Quantity`

Y plane component value.

Attribute detail

`Point2D.flat`

Method detail

`Point2D.__eq__(other: Point2D)` → `bool`

Equals operator for the `Point2D` class.

`Point2D.__ne__(other: Point2D)` → `bool`

Not equals operator for the `Point2D` class.

`Point2D.__add__(other: Point2D | ansys.geometry.core.math.vector.Vector2D)` → `Point2D`

Add operation for the `Point2D` class.

`Point2D.__sub__(other: Point2D)` → `Point2D`

Subtraction operation for the `Point2D` class.

`Point2D.unit()` → `pint.Unit`

Get the unit of the object.

`Point2D.base_unit()` → `pint.Unit`

Get the base unit of the object.

`Point3D`

```
class ansys.geometry.core.math.point.Point3D(input: numpy.ndarray |
                                              ansys.geometry.core.typing.RealSequence =
                                              DEFAULT_POINT3D_VALUES, unit: pint.Unit | None =
                                              None)
```

Bases: `numpy.ndarray`, `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides geometry primitive representation for a 3D point.

Parameters

input

[`ndarray` | `RealSequence`, default: `DEFAULT_POINT3D_VALUES`] Direction arguments, either as a `numpy.ndarray` class or as a `RealSequence`.

unit

[`Unit` | `None`, default: `DEFAULT_UNITS.LENGTH`] Units for defining 3D point values. If not specified, the default unit is `DEFAULT_UNITS.LENGTH`.

Overview

Methods

<code>unit</code>	Get the unit of the object.
<code>base_unit</code>	Get the base unit of the object.
<code>transform</code>	Transform the 3D point with a transformation matrix.

Properties

<code>x</code>	X plane component value.
<code>y</code>	Y plane component value.
<code>z</code>	Z plane component value.
<code>position</code>	Get the position of the point as a numpy array.

Attributes

`flat`

Special methods

<code>__eq__</code>	Equals operator for the <code>Point3D</code> class.
<code>__ne__</code>	Not equals operator for the <code>Point3D</code> class.
<code>__add__</code>	Add operation for the <code>Point3D</code> class.
<code>__sub__</code>	Subtraction operation for the <code>Point3D</code> class.

Import detail

```
from ansys.geometry.core.math.point import Point3D
```

Property detail

property `Point3D.x`: `pint.Quantity`

X plane component value.

property `Point3D.y`: `pint.Quantity`

Y plane component value.

property `Point3D.z`: `pint.Quantity`

Z plane component value.

property `Point3D.position`: `numpy.ndarray`

Get the position of the point as a numpy array.

Attribute detail

`Point3D.flat`

Method detail

`Point3D.__eq__(other: Point3D) → bool`

Equals operator for the `Point3D` class.

`Point3D.__ne__(other: Point3D) → bool`

Not equals operator for the `Point3D` class.

`Point3D.__add__(other: Point3D | ansys.geometry.core.math.vector.Vector3D) → Point3D`

Add operation for the `Point3D` class.

`Point3D.__sub__(other: Point3D | ansys.geometry.core.math.vector.Vector3D) → Point3D`

Subtraction operation for the `Point3D` class.

`Point3D.unit()` → `pint.Unit`

Get the unit of the object.

`Point3D.base_unit()` → `pint.Unit`

Get the base unit of the object.

`Point3D.transform(matrix: ansys.geometry.core.math.matrix.Matrix44) → Point3D`

Transform the 3D point with a transformation matrix.

Parameters

matrix

[`Matrix44`] 4x4 transformation matrix to apply to the point.

Returns

Point3D

New 3D point that is the transformed copy of the original 3D point after applying the transformation matrix.

Notes

Transform the `Point3D` object by applying the specified 4x4 transformation matrix and return a new `Point3D` object representing the transformed point.

Description

Provides geometry primitive representation for 2D and 3D points.

Module detail

`ansys.geometry.core.math.point.DEFAULT_POINT2D_VALUES`

Default values for a 2D point.

`ansys.geometry.core.math.point.DEFAULT_POINT3D_VALUES`

Default values for a 3D point.

`ansys.geometry.core.math.point.BASE_UNIT_LENGTH`

Default value for the length of the base unit.

The `vector.py` module

Summary

Classes

<code>Vector3D</code>	Provides for managing and creating a 3D vector.
<code>Vector2D</code>	Provides for creating and managing a 2D vector.
<code>UnitVector3D</code>	Provides for creating and managing a 3D unit vector.
<code>UnitVector2D</code>	Provides for creating and managing a 3D unit vector.

Vector3D

class `ansys.geometry.core.math.vector.Vector3D`(*shape, dtype=float, buffer=None, offset=0, strides=None, order=None*)

Bases: `numpy.ndarray`

Provides for managing and creating a 3D vector.

Parameters

input

[`ndarray` | `RealSequence`] 3D `numpy.ndarray` class with shape(X,).

Overview

Constructors

<code>from_points</code>	Create a 3D vector from two distinct 3D points.
--------------------------	---

Methods

<code>is_perpendicular_to</code>	Check if this vector and another vector are perpendicular.
<code>is_parallel_to</code>	Check if this vector and another vector are parallel.
<code>is_opposite</code>	Check if this vector and another vector are opposite.
<code>normalize</code>	Return a normalized version of the 3D vector.
<code>transform</code>	Transform the 3D vector3D with a transformation matrix.
<code>rotate_vector</code>	Rotate a vector around a given axis by a specified angle.
<code>get_angle_between</code>	Get the angle between this 3D vector and another 3D vector.
<code>cross</code>	Return the cross product of Vector3D objects.

Properties

<code>x</code>	X coordinate of the Vector3D class.
<code>y</code>	Y coordinate of the Vector3D class.
<code>z</code>	Z coordinate of the Vector3D class.
<code>norm</code>	Norm of the vector.
<code>magnitude</code>	Norm of the vector.
<code>is_zero</code>	Check if all components of the 3D vector are zero.

Special methods

<code>__eq__</code>	Equals operator for the Vector3D class.
<code>__ne__</code>	Not equals operator for the Vector3D class.
<code>__mul__</code>	Overload * operator with dot product.
<code>__mod__</code>	Overload % operator with cross product.
<code>__add__</code>	Addition operation overload for 3D vectors.
<code>__sub__</code>	Subtraction operation overload for 3D vectors.

Import detail

```
from ansys.geometry.core.math.vector import Vector3D
```

Property detail

property Vector3D.**x**: `ansys.geometry.core.typing.Real`
X coordinate of the Vector3D class.

property Vector3D.**y**: `ansys.geometry.core.typing.Real`
Y coordinate of the Vector3D class.

property Vector3D.**z**: `ansys.geometry.core.typing.Real`
Z coordinate of the Vector3D class.

property Vector3D.**norm**: `float`
Norm of the vector.

property Vector3D.**magnitude**: `float`
Norm of the vector.

property `Vector3D.is_zero`: `bool`

Check if all components of the 3D vector are zero.

Method detail

`Vector3D.is_perpendicular_to(other_vector: Vector3D) → bool`

Check if this vector and another vector are perpendicular.

`Vector3D.is_parallel_to(other_vector: Vector3D) → bool`

Check if this vector and another vector are parallel.

`Vector3D.is_opposite(other_vector: Vector3D) → bool`

Check if this vector and another vector are opposite.

`Vector3D.normalize() → Vector3D`

Return a normalized version of the 3D vector.

`Vector3D.transform(matrix: ansys.geometry.core.math.matrix.Matrix44) → Vector3D`

Transform the 3D vector3D with a transformation matrix.

Parameters

matrix

[Matrix44] 4x4 transformation matrix to apply to the vector.

Returns

Vector3D

A new 3D vector that is the transformed copy of the original 3D vector after applying the transformation matrix.

Notes

Transform the `Vector3D` object by applying the specified 4x4 transformation matrix and return a new `Vector3D` object representing the transformed vector.

`Vector3D.rotate_vector(vector: Vector3D, angle: ansys.geometry.core.typing.Real | pint.Quantity | ansys.geometry.core.misc.measurements.Angle) → Vector3D`

Rotate a vector around a given axis by a specified angle.

`Vector3D.get_angle_between(v: Vector3D) → pint.Quantity`

Get the angle between this 3D vector and another 3D vector.

Parameters

v

[Vector3D] Other 3D vector for computing the angle.

Returns

Quantity

Angle between these two 3D vectors.

`Vector3D.cross(v: Vector3D) → Vector3D`

Return the cross product of `Vector3D` objects.

`Vector3D.__eq__(other: Vector3D) → bool`

Equals operator for the `Vector3D` class.

`Vector3D.__ne__(other: Vector3D) → bool`

Not equals operator for the Vector3D class.

`Vector3D.__mul__(other: Vector3D | ansys.geometry.core.typing.Real) → Vector3D | ansys.geometry.core.typing.Real`

Overload * operator with dot product.

Notes

This method also admits scalar multiplication.

`Vector3D.__mod__(other: Vector3D) → Vector3D`

Overload % operator with cross product.

`Vector3D.__add__(other: Vector3D | ansys.geometry.core.math.point.Point3D) → Vector3D | ansys.geometry.core.math.point.Point3D`

Addition operation overload for 3D vectors.

`Vector3D.__sub__(other: Vector3D) → Vector3D`

Subtraction operation overload for 3D vectors.

classmethod `Vector3D.from_points(point_a: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D, point_b: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point3D)`

Create a 3D vector from two distinct 3D points.

Parameters

point_a

[`ndarray` | `RealSequence` | `Point3D`] `Point3D` class representing the first point.

point_b

[`ndarray` | `RealSequence` | `Point3D`] `Point3D` class representing the second point.

Returns

Vector3D

3D vector from point_a to point_b.

Notes

The resulting 3D vector is always expressed in Point3D base units.

Vector2D

class `ansys.geometry.core.math.vector.Vector2D(shape, dtype=float, buffer=None, offset=0, strides=None, order=None)`

Bases: `numpy.ndarray`

Provides for creating and managing a 2D vector.

Parameters

input

[`ndarray` | `RealSequence`] 2D `numpy.ndarray` class with shape(X,).

Overview

Constructors

<code>from_points</code>	Create a 2D vector from two distinct 2D points.
--------------------------	---

Methods

<code>cross</code>	Return the cross product of Vector2D objects.
<code>is_perpendicular_to</code>	Check if this 2D vector and another 2D vector are perpendicular.
<code>is_parallel_to</code>	Check if this vector and another vector are parallel.
<code>is_opposite</code>	Check if this vector and another vector are opposite.
<code>normalize</code>	Return a normalized version of the 2D vector.
<code>get_angle_between</code>	Get the angle between this 2D vector and another 2D vector.

Properties

<code>x</code>	X coordinate of the 2D vector.
<code>y</code>	Y coordinate of the 2D vector.
<code>norm</code>	Norm of the 2D vector.
<code>magnitude</code>	Norm of the 2D vector.
<code>is_zero</code>	Check if values for all components of the 2D vector are zero.

Special methods

<code>__eq__</code>	Equals operator for the 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) → ansys.geometry.core.typing.Real`

Return the cross product of `Vector2D` objects.

`Vector2D.is_perpendicular_to(other_vector: Vector2D) → bool`

Check if this 2D vector and another 2D vector are perpendicular.

`Vector2D.is_parallel_to(other_vector: Vector2D) → bool`

Check if this vector and another vector are parallel.

`Vector2D.is_opposite(other_vector: Vector2D) → bool`

Check if this vector and another vector are opposite.

`Vector2D.normalize() → Vector2D`

Return a normalized version of the 2D vector.

`Vector2D.get_angle_between(v: Vector2D) → pint.Quantity`

Get the angle between this 2D vector and another 2D vector.

Parameters

v

[`Vector2D`] Other 2D vector to compute the angle with.

Returns

Quantity

Angle between these two 2D vectors.

`Vector2D.__eq__(other: Vector2D) → bool`

Equals operator for the `Vector2D` class.

`Vector2D.__ne__(other: Vector2D) → bool`

Not equals operator for the `Vector2D` class.

`Vector2D.__mul__(other: Vector2D | ansys.geometry.core.typing.Real) → Vector2D | ansys.geometry.core.typing.Real`

Overload `*` operator with dot product.

Notes

This method also admits scalar multiplication.

`Vector2D.__add__(other: Vector2D | ansys.geometry.core.math.point.Point2D) → Vector2D | ansys.geometry.core.math.point.Point2D`

Addition operation overload for 2D vectors.

`Vector2D.__sub__(other: Vector2D) → Vector2D`

Subtraction operation overload for 2D vectors.

`Vector2D.__mod__(other: Vector2D) → ansys.geometry.core.typing.Real`

Overload `%` operator with cross product.

```
classmethod Vector2D.from_points(point_a: numpy.ndarray | ansys.geometry.core.typing.RealSequence |  
                                ansys.geometry.core.math.point.Point2D, point_b: numpy.ndarray |  
                                ansys.geometry.core.typing.RealSequence |  
                                ansys.geometry.core.math.point.Point2D)
```

Create a 2D vector from two distinct 2D points.

Parameters

point_a

[*ndarray* | RealSequence | Point2D] *Point2D* class representing the first point.

point_b

[*ndarray* | RealSequence | Point2D] *Point2D* class representing the second point.

Returns

Vector2D

2D vector from point_a to point_b.

Notes

The resulting 2D vector is always expressed in *Point2D* base units.

UnitVector3D

```
class ansys.geometry.core.math.vector.UnitVector3D(shape, dtype=float, buffer=None, offset=0,  
                                                    strides=None, order=None)
```

Bases: *Vector3D*

Provides for creating and managing a 3D unit vector.

Parameters

input

[*ndarray* | RealSequence | *Vector3D*]

- 1D *numpy.ndarray* class with shape(X,)
- *Vector3D*

Overview

Constructors

<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: numpy.ndarray | ansys.geometry.core.typing.RealSequence  
| ansys.geometry.core.math.point.Point3D, point_b: numpy.ndarray |  
ansys.geometry.core.typing.RealSequence |  
ansys.geometry.core.math.point.Point3D)
```

Create a 3D unit vector from two distinct 3D points.

Parameters**point_a**

[[ndarray](#) | RealSequence | Point3D] [Point3D](#) class representing the first point.

point_b

[[ndarray](#) | RealSequence | Point3D] [Point3D](#) class representing the second point.

Returns**UnitVector3D**

3D unit vector from point_a to point_b.

UnitVector2D

class ansys.geometry.core.math.vector.**UnitVector2D**(*shape, dtype=float, buffer=None, offset=0, strides=None, order=None*)

Bases: Vector2D

Provides for creating and managing a 3D unit vector.

Parameters**input**

[[ndarray](#) | RealSequence | Vector2D]

- 1D [numpy.ndarray](#) class with shape(X,)
- Vector2D

Overview**Constructors**

[from_points](#) Create a 2D unit vector from two distinct 2D points.

Import detail

```
from ansys.geometry.core.math.vector import UnitVector2D
```

Method detail

classmethod UnitVector2D.**from_points**(*point_a: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point2D, point_b: numpy.ndarray | ansys.geometry.core.typing.RealSequence | ansys.geometry.core.math.point.Point2D*)

Create a 2D unit vector from two distinct 2D points.

Parameters**point_a**

[[ndarray](#) | RealSequence | Point2D] [Point2D](#) class representing the first point.

point_b

[[ndarray](#) | RealSequence | Point2D] [Point2D](#) class representing the second point.

Returns

UnitVector2D

2D unit vector from point_a to point_b.

Description

Provides for creating and managing 2D and 3D vectors.

Description

PyAnsys Geometry math subpackage.

The misc package**Summary****Submodules**

<i>accuracy</i>	Provides for evaluating decimal precision.
<i>auxiliary</i>	Auxiliary functions for the PyAnsys Geometry library.
<i>checks</i>	Provides functions for performing common checks.
<i>measurements</i>	Provides various measurement-related classes.
<i>options</i>	Provides various option classes.
<i>units</i>	Provides for handling units homogeneously throughout PyAnsys Geometry.

The accuracy.py module**Summary****Classes**

<i>Accuracy</i>	Decimal precision evaluations for math operations.
-----------------	--

Constants

<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
```

Decimal precision evaluations for math operations.

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 length is greater than the reference length.
<code>length_is_less_than_or_equal</code>	Check if the length is less than or equal to the reference length.
<code>length_is_zero</code>	Check if the length is within the length accuracy of exact zero.
<code>length_is_negative</code>	Check if the length is below a negative length accuracy.
<code>length_is_positive</code>	Check if the length is above a positive length accuracy.
<code>angle_is_zero</code>	Check if the length is within the angle accuracy of exact zero.
<code>angle_is_negative</code>	Check if the angle is below a negative angle accuracy.
<code>angle_is_positive</code>	Check if the angle is above a positive angle accuracy.
<code>is_within_tolerance</code>	Check if two values are inside a relative and absolute tolerance.

Import detail

```
from ansys.geometry.core.misc.accuracy import Accuracy
```

Method detail

static `Accuracy.length_accuracy()` → *ansys.geometry.core.typing.Real*

Return the LENGTH_ACCURACY constant.

static `Accuracy.angle_accuracy()` → *ansys.geometry.core.typing.Real*

Return the ANGLE_ACCURACY constant.

static `Accuracy.double_accuracy()` → *ansys.geometry.core.typing.Real*

Return the DOUBLE_ACCURACY constant.

static `Accuracy.length_is_equal(comparison_length: ansys.geometry.core.typing.Real, reference_length: ansys.geometry.core.typing.Real)` → `bool`

Check if the comparison length is equal to the reference length.

Returns

bool

True if the comparison length is equal to the reference length within the length accuracy, False otherwise.

Notes

The check is done up to the constant value specified for LENGTH_ACCURACY.

static `Accuracy.equal_doubles(a: ansys.geometry.core.typing.Real, b: ansys.geometry.core.typing.Real)`

Compare two double values.

static `Accuracy.compare_with_tolerance(a: ansys.geometry.core.typing.Real, b: ansys.geometry.core.typing.Real, relative_tolerance: ansys.geometry.core.typing.Real, absolute_tolerance: ansys.geometry.core.typing.Real)` → *ansys.geometry.core.typing.Real*

Compare two doubles given the relative and absolute tolerances.

```
static Accuracy.length_is_greater_than_or_equal(comparison_length: ansys.geometry.core.typing.Real,  
                                              reference_length: ansys.geometry.core.typing.Real) → bool
```

Check if the length is greater than the reference length.

Returns

bool

True if the comparison length is greater than the reference length within the length accuracy,
False otherwise.

Notes

The check is done up to the constant value specified for LENGTH_ACCURACY.

```
static Accuracy.length_is_less_than_or_equal(comparison_length: ansys.geometry.core.typing.Real,  
                                           reference_length: ansys.geometry.core.typing.Real) → bool
```

Check if the length is less than or equal to the reference length.

Returns

bool

True if the comparison length is less than or equal to the reference length within the length
accuracy, False otherwise.

Notes

The check is done up to the constant value specified for LENGTH_ACCURACY.

```
static Accuracy.length_is_zero(length: ansys.geometry.core.typing.Real) → bool
```

Check if the length is within the length accuracy of exact zero.

Returns

bool

True if the length is within the length accuracy of exact zero, False otherwise.

```
static Accuracy.length_is_negative(length: ansys.geometry.core.typing.Real) → bool
```

Check if the length is below a negative length accuracy.

Returns

bool

True if the length is below a negative length accuracy,
False otherwise.

```
static Accuracy.length_is_positive(length: ansys.geometry.core.typing.Real) → bool
```

Check if the length is above a positive length accuracy.

Returns

bool

True if the length is above a positive length accuracy,
False otherwise.

static Accuracy.**angle_is_zero**(*angle*: ansys.geometry.core.typing.Real) → bool

Check if the length is within the angle accuracy of exact zero.

Returns

bool

True if the length is within the angle accuracy of exact zero,
False otherwise.

static Accuracy.**angle_is_negative**(*angle*: ansys.geometry.core.typing.Real) → bool

Check if the angle is below a negative angle accuracy.

Returns

bool

True if the angle is below a negative angle accuracy,
False otherwise.

static Accuracy.**angle_is_positive**(*angle*: ansys.geometry.core.typing.Real) → bool

Check if the angle is above a positive angle accuracy.

Returns

bool

True if the angle is above a positive angle accuracy,
False otherwise.

static Accuracy.**is_within_tolerance**(*a*: ansys.geometry.core.typing.Real, *b*:
ansys.geometry.core.typing.Real, *relative_tolerance*:
ansys.geometry.core.typing.Real, *absolute_tolerance*:
ansys.geometry.core.typing.Real) → bool

Check if two values are inside a relative and absolute tolerance.

Parameters

a

[Real] First value.

b

[Real] Second value.

relative_tolerance

[Real] Relative tolerance accepted.

absolute_tolerance

[Real] Absolute tolerance accepted.

Returns

bool

True if the values are inside the accepted tolerances, False otherwise.

Description

Provides for evaluating decimal precision.

Module detail

`ansys.geometry.core.misc.accuracy.LENGTH_ACCURACY`: *ansys.geometry.core.typing.Real* = **1e-08**

Constant for decimal accuracy in length comparisons.

`ansys.geometry.core.misc.accuracy.ANGLE_ACCURACY`: *ansys.geometry.core.typing.Real* = **1e-06**

Constant for decimal accuracy in angle comparisons.

`ansys.geometry.core.misc.accuracy.DOUBLE_ACCURACY`: *ansys.geometry.core.typing.Real* = **1e-13**

Constant for double accuracy.

The auxiliary.py module

Summary

Functions

<i>get_design_from_component</i>	Get the Design of the given Component object.
<i>get_design_from_body</i>	Get the Design of the given Body object.
<i>get_design_from_face</i>	Get the Design of the given Face object.
<i>get_design_from_edge</i>	Get the Design of the given Edge object.
<i>get_all_bodies_from_design</i>	Find all the Body objects inside a Design.
<i>get_bodies_from_ids</i>	Find the Body objects inside a Design from its ids.
<i>get_components_from_ids</i>	Find the Component objects inside a Design from its ids.
<i>get_faces_from_ids</i>	Find the Face objects inside a Design from its ids.
<i>get_edges_from_ids</i>	Find the Edge objects inside a Design from its ids.
<i>get_vertices_from_ids</i>	Find the Vertex objects inside a Design from its ids.
<i>get_beams_from_ids</i>	Find the Beam objects inside a Design from its ids.
<i>get_design_points_from_ids</i>	Find the DesignPoint objects inside a Design from its ids.
<i>convert_color_to_hex</i>	Get the hex string color from input formats.
<i>convert_opacity_to_hex</i>	Get the hex string from an opacity value.
<i>prepare_file_for_server_upload</i>	Create a zip file from the given file path.

Constants

<i>DEFAULT_COLOR</i>

Description

Auxiliary functions for the PyAnsys Geometry library.

Module detail

`ansys.geometry.core.misc.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.

```
ansys.geometry.core.misc.auxiliary.get_design_from_body(body:
    ansys.geometry.core.designer.body.Body) → an-
    sys.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.

```
ansys.geometry.core.misc.auxiliary.get_design_from_face(face:
    ansys.geometry.core.designer.face.Face) →
    an-
    sys.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.

```
ansys.geometry.core.misc.auxiliary.get_design_from_edge(edge:
    ansys.geometry.core.designer.edge.Edge) →
    an-
    sys.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.

```
ansys.geometry.core.misc.auxiliary.get_all_bodies_from_design(design: an-
    sys.geometry.core.designer.design.Design) →
    list[ansys.geometry.core.designer.body.Body]
```

Find all the Body objects inside a Design.

Parameters**design**

[Design] Parent design for the bodies.

Returns

`list[Body]`

List of Body objects.

Notes

This method takes a design and gets the corresponding Body objects.

```
ansys.geometry.core.misc.auxiliary.get_bodies_from_ids(design:
ansys.geometry.core.designer.design.Design,
body_ids: list[str]) →
list[ansys.geometry.core.designer.body.Body]
```

Find the Body objects inside a Design from its ids.

Parameters**design**

[Design] Parent design for the faces.

body_ids

[`list[str]`] List of body ids.

Returns

`list[Body]`

List of Body objects.

Notes

This method takes a design and body ids, and gets their corresponding Body object.

```
ansys.geometry.core.misc.auxiliary.get_components_from_ids(design: an-
sys.geometry.core.designer.design.Design,
component_ids: list[str]) →
list[ansys.geometry.core.designer.component.Component]
```

Find the Component objects inside a Design from its ids.

Parameters**design**

[Design] Parent design for the components.

component_ids

[`list[str]`] List of component ids.

Returns

`list[Component]`

List of Component objects.

Notes

This method takes a design and component ids, and gets their corresponding Component object.

```
ansys.geometry.core.misc.auxiliary.get_faces_from_ids(design:
    ansys.geometry.core.designer.design.Design,
    face_ids: list[str]) →
    list[ansys.geometry.core.designer.face.Face]
```

Find the Face objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the faces.

face_ids
[list[str]] List of face ids.

Returns

list[Face]
List of Face objects.

Notes

This method takes a design and face ids, and gets their corresponding Face object.

```
ansys.geometry.core.misc.auxiliary.get_edges_from_ids(design:
    ansys.geometry.core.designer.design.Design,
    edge_ids: list[str]) →
    list[ansys.geometry.core.designer.edge.Edge]
```

Find the Edge objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the edges.

edge_ids
[list[str]] List of edge ids.

Returns

list[Edge]
List of Edge objects.

Notes

This method takes a design and edge ids, and gets their corresponding Edge objects.

```
ansys.geometry.core.misc.auxiliary.get_vertices_from_ids(design: an-
    sys.geometry.core.designer.design.Design,
    vertex_ids: list[str]) →
    list[ansys.geometry.core.designer.vertex.Vertex]
```

Find the Vertex objects inside a Design from its ids.

Parameters

design
[Design] Parent design for the vertices.

vertex_ids
[list[str]] List of vertex ids.

Returns

`list[Vertex]`

List of Vertex objects.

Notes

This method takes a design and vertex ids, and gets their corresponding Vertex objects.

```
ansys.geometry.core.misc.auxiliary.get_beams_from_ids(design:
                                                    ansys.geometry.core.designer.design.Design,
                                                    beam_ids: list[str]) →
                                                    list[ansys.geometry.core.designer.beam.Beam]
```

Find the Beam objects inside a Design from its ids.

Parameters

design

[Design] Parent design for the beams.

beam_ids

[list[str]] List of beam ids.

Returns

`list[Beam]`

List of Beam objects.

Notes

This method takes a design and beam ids, and gets their corresponding Beam objects.

```
ansys.geometry.core.misc.auxiliary.get_design_points_from_ids(design: an-
                                                              sys.geometry.core.designer.design.Design,
                                                              design_point_ids: list[str]) →
                                                              list[ansys.geometry.core.designer.designpoint.DesignPoint]
```

Find the DesignPoint objects inside a Design from its ids.

Parameters

design

[Design] Parent design for the design points.

design_point_ids

[list[str]] List of design point ids.

Returns

`list[DesignPoint]`

List of DesignPoint objects.

Notes

This method takes a design and design point ids, and gets their corresponding DesignPoint objects.

```
ansys.geometry.core.misc.auxiliary.convert_color_to_hex(color: str | tuple[float, float, float] |
                                                         tuple[float, float, float, float]) → str
```

Get the hex string color from input formats.

Parameters

color

[str | tuple[float, float, float] | tuple[float, float, float, float]] Color to set

the body to. This can be a string representing a color name or a tuple of RGB values in the range [0, 1] (RGBA) or [0, 255] (pure RGB).

Returns

str

The hex code string for the color, formatted #rrggbbaa.

`ansys.geometry.core.misc.auxiliary.convert_opacity_to_hex(opacity: float) → str`

Get the hex string from an opacity value.

Parameters

opacity

[float] Opacity to set body to. Must be in the range [0, 1].

Returns

The hex code for the opacity formatted #aa

`ansys.geometry.core.misc.auxiliary.prepare_file_for_server_upload(file_path: pathlib.Path) → pathlib.Path`

Create a zip file from the given file path.

Parameters

file_path

[str] The path to the file to be zipped.

Returns

Path

The path to the created zip file.

`ansys.geometry.core.misc.auxiliary.DEFAULT_COLOR = '#D6F7D1'`

The checks.py module

Summary

Functions

<code>ensure_design_is_active</code>	Make sure that the design is active before executing a method.
<code>check_is_float_int</code>	Check if a parameter has a float or integer value.
<code>check_ndarray_is_float_int</code>	Check if a <code>numpy.ndarray</code> has float/integer types.
<code>check_ndarray_is_not_none</code>	Check if a <code>numpy.ndarray</code> is all None.
<code>check_ndarray_is_all_nan</code>	Check if a <code>numpy.ndarray</code> is all nan-valued.
<code>check_ndarray_is_non_zero</code>	Check if a <code>numpy.ndarray</code> is zero-valued.
<code>check_pint_unit_compatibility</code>	Check if input <code>pint.Unit</code> is compatible with the expected input.
<code>check_type_equivalence</code>	Check if an input object is of the same class as an expected object.
<code>check_type</code>	Check if an input object is of the same type as expected types.
<code>check_type_all_elements_in_iterable</code>	Check if all elements in an iterable are of the same type as expected.
<code>check_nurbs_compatibility</code>	Check if the inputs require NURBS functionality and it is available.
<code>min_backend_version</code>	Compare a minimum required version to the current backend version.
<code>deprecated_method</code>	Decorate a method as deprecated.
<code>deprecated_argument</code>	Decorate a method argument as deprecated.
<code>run_if_graphics_required</code>	Check if graphics are available.
<code>graphics_required</code>	Decorate a method as requiring graphics.
<code>kwargs_passed_not_accepted</code>	Check that no unexpected kwargs are passed to the method.

Constants

<code>ERROR_GRAPHICS_REQUIRED</code>	Message to display when graphics are required for a method.
--------------------------------------	---

Description

Provides functions for performing common checks.

Module detail

`ansys.geometry.core.misc.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.

`ansys.geometry.core.misc.checks.check_is_float_int(param: object, param_name: str | None = None) → None`

Check if a parameter has a float or integer value.

Parameters

`param`

[`object`] Object instance to check.

`param_name`

[`str`, default: `None`] Parameter name (if any).

Raises

`TypeError`

If the parameter does not have a float or integer value.

`ansys.geometry.core.misc.checks.check_ndarray_is_float_int`(*param: numpy.ndarray, param_name: str | None = None*) → *None*

Check if a `numpy.ndarray` has float/integer types.

Parameters

param

[`ndarray`] `numpy.ndarray` instance to check.

param_name

[`str`, default: `None`] `numpy.ndarray` instance name (if any).

Raises

TypeError

If the `numpy.ndarray` instance does not have float or integer values.

`ansys.geometry.core.misc.checks.check_ndarray_is_not_none`(*param: numpy.ndarray, param_name: str | None = None*) → *None*

Check if a `numpy.ndarray` is all `None`.

Parameters

param

[`ndarray`] `numpy.ndarray` instance to check.

param_name

[`str`, default: `None`] `numpy.ndarray` instance name (if any).

Raises

ValueError

If the `numpy.ndarray` instance has a value of `None` for all parameters.

`ansys.geometry.core.misc.checks.check_ndarray_is_all_nan`(*param: numpy.ndarray, param_name: str | None = None*) → *None*

Check if a `numpy.ndarray` is all nan-valued.

Parameters

param

[`ndarray`] `numpy.ndarray` instance to check.

param_name

[`str` or `None`, default: `None`] `numpy.ndarray` instance name (if any).

Raises

ValueError

If the `numpy.ndarray` instance is all nan-valued.

`ansys.geometry.core.misc.checks.check_ndarray_is_non_zero`(*param: numpy.ndarray, param_name: str | None = None*) → *None*

Check if a `numpy.ndarray` is zero-valued.

Parameters

param

[`ndarray`] `numpy.ndarray` instance to check.

param_name

[`str`, default: `None`] `numpy.ndarray` instance name (if any).

Raises

ValueError

If the `numpy.ndarray` instance is zero-valued.

`ansys.geometry.core.misc.checks.check_pint_unit_compatibility(input: pint.Unit, expected: pint.Unit) → None`

Check if input `pint.Unit` is compatible with the expected input.

Parameters**input**

[Unit] `pint.Unit` input.

expected

[Unit] `pint.Unit` expected dimensionality.

Raises**TypeError**

If the input is not compatible with the `pint.Unit` class.

`ansys.geometry.core.misc.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.

`ansys.geometry.core.misc.checks.check_type(input: object, expected_type: Type | tuple[Type, Ellipsis]) → None`

Check if an input object is of the same type as expected types.

Parameters**input**

[object] Input object.

expected_type

[Type | tuple[Type, ...]] One or more types to compare the input object against.

Raises**TypeError**

If the object does not match the one or more expected types.

`ansys.geometry.core.misc.checks.check_type_all_elements_in_iterable(input: collections.abc.Iterable, expected_type: Type | tuple[Type, Ellipsis]) → None`

Check if all elements in an iterable are of the same type as expected.

Parameters

input

[Iterable] Input iterable.

expected_type

[Type | tuple[Type, ...]] One or more types to compare the input object against.

Raises

TypeError

If one of the elements in the iterable does not match the one or more expected types.

```
ansys.geometry.core.misc.checks.check_nurbs_compatibility(backend_version: semver.Version,
                                                         sketch: an-
                                                         sys.geometry.core.sketch.sketch.Sketch
                                                         = None, curves:
                                                         list[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve]
                                                         = None, surfaces:
                                                         list[ansys.geometry.core.shapes-surfaces.trimmed_surface.TrimmedSurface]
                                                         = None) → None
```

Check if the inputs require NURBS functionality and it is available.

Parameters

backend_version

[semver.Version] Backend version to check against.

sketch

[Sketch, default: None] Sketch to check for NURBS geometry.

curves

[list[TrimmedCurve], default: None] List of TrimmedCurve to check for NURBS geometry.

surfaces

[list[TrimmedSurface], default: None] List of TrimmedSurface to check for NURBS geometry.

Raises

GeometryRuntimeError

If inputs contain NURBS functionality but the backend is older than 26R1.

```
ansys.geometry.core.misc.checks.min_backend_version(major: int, minor: int, service_pack: int)
```

Compare a minimum required version to the current backend version.

Parameters

major

[int] Minimum major version required by the method.

minor

[int] Minimum minor version required by the method.

service_pack

[int] Minimum service pack version required by the method.

Raises

GeometryRuntimeError

If the method version is higher than the backend version.

GeometryRuntimeError

If the client is not available.

`ansys.geometry.core.misc.checks.deprecated_method`(*alternative: str | None = None, info: str | None = None, version: str | None = None, remove: str | None = None*)

Decorate a method as deprecated.

Parameters

alternative

[*str*, default: *None*] Alternative method to use. If provided, the warning message will include the alternative method.

info

[*str*, default: *None*] Additional information to include in the warning message.

version

[*str*, default: *None*] Version where the method was deprecated.

remove

[*str*, default: *None*] Version where the method will be removed.

`ansys.geometry.core.misc.checks.deprecated_argument`(*arg: str, alternative: str | None = None, info: str | None = None, version: str | None = None, remove: str | None = None*)

Decorate a method argument as deprecated.

Parameters

arg

[*str*] Argument to mark as deprecated.

alternative

[*str*, default: *None*] Alternative argument to use. If provided, the warning message will include the alternative argument.

info

[*str*, default: *None*] Additional information to include in the warning message.

version

[*str*, default: *None*] Version where the method was deprecated.

remove

[*str*, default: *None*] Version where the method will be removed.

`ansys.geometry.core.misc.checks.run_if_graphics_required`()

Check if graphics are available.

`ansys.geometry.core.misc.checks.graphics_required`(*method*)

Decorate a method as requiring graphics.

Parameters

method

[*callable*()] Method to decorate.

Returns

callable()

Decorated method.

`ansys.geometry.core.misc.checks.kwargs_passed_not_accepted`(*method*)

Check that no unexpected kwargs are passed to the method.

This decorator will raise a `TypeError` if any keyword arguments are passed to the decorated method that don't correspond to the method's parameters. If the method has `**kwargs` in its signature, this decorator will raise an error for any kwargs passed to it (that are not explicitly accepted).

Parameters

method

[`callable()`] The method to decorate.

Returns

`callable()`

Decorated method that raises `TypeError` if unexpected kwargs are passed.

Raises

`TypeError`

If unexpected keyword arguments are passed to the decorated method.

Examples

```
>>> @kwargs_passed_not_accepted
... def my_method(arg1, arg2):
...     return arg1 + arg2
>>> my_method(1, 2) # Works fine
3
>>> my_method(arg1=1, arg2=2) # Works fine
3
>>> my_method(1, 2, invalid_arg=3) # Raises TypeError
TypeError: The following keyword arguments are not accepted
in the method 'my_method': invalid_arg.
>>> @kwargs_passed_not_accepted
... def my_method_with_kwargs(arg1, arg2, **kwargs):
...     return arg1 + arg2
>>> my_method_with_kwargs(1, 2, invalid_arg=3) # Raises TypeError
TypeError: The following keyword arguments are not accepted
in the method 'my_method_with_kwargs': invalid_arg.
```

`ansys.geometry.core.misc.checks.ERROR_GRAPHICS_REQUIRED = 'Graphics are required for this method. Please install the ``graphics`` target to use this...'`

Message to display when graphics are required for a method.

The `measurements.py` module

Summary

Classes

<code>SingletonMeta</code>	Provides a thread-safe implementation of a singleton design pattern.
<code>DefaultUnitsClass</code>	Provides default units for PyAnsys Geometry.
<code>Measurement</code>	Provides the <code>PhysicalQuantity</code> subclass for holding a measurement.
<code>Distance</code>	Provides the <code>Measurement</code> subclass for holding a distance.
<code>Angle</code>	Provides the <code>Measurement</code> subclass for holding an angle.
<code>Area</code>	Provides the <code>Measurement</code> subclass for holding an area.

Constants

<code>DEFAULT_UNITS</code>	PyAnsys Geometry default units object.
----------------------------	--

SingletonMeta

class ansys.geometry.core.misc.measurements.SingletonMeta

Bases: `type`

Provides a thread-safe implementation of a singleton design pattern.

Overview

Special methods

<code>__call__</code>	Return a single instance of the class.
-----------------------	--

Import detail

```
from ansys.geometry.core.misc.measurements import SingletonMeta
```

Method detail

SingletonMeta.__call__(*args, **kwargs)

Return a single instance of the class.

Possible changes to the value of the `__init__` argument do not affect the returned instance.

DefaultUnitsClass

class ansys.geometry.core.misc.measurements.DefaultUnitsClass

Provides default units for PyAnsys Geometry.

Overview

Properties

<code>LENGTH</code>	Default length unit for PyAnsys Geometry.
<code>ANGLE</code>	Default angle unit for PyAnsys Geometry.
<code>AREA</code>	Default area unit for PyAnsys Geometry.
<code>VOLUME</code>	Default volume unit for PyAnsys Geometry.
<code>SERVER_LENGTH</code>	Default length unit for gRPC messages.
<code>SERVER_AREA</code>	Default area unit for gRPC messages.
<code>SERVER_VOLUME</code>	Default volume unit for gRPC messages.
<code>SERVER_ANGLE</code>	Default angle unit for gRPC messages.

Import detail

```
from ansys.geometry.core.misc.measurements import DefaultUnitsClass
```

Property detail

property DefaultUnitsClass.LENGTH: `pint.Unit`

Default length unit for PyAnsys Geometry.

property DefaultUnitsClass.ANGLE: `pint.Unit`

Default angle unit for PyAnsys Geometry.

property DefaultUnitsClass.AREA: `pint.Unit`

Default area unit for PyAnsys Geometry.

property DefaultUnitsClass.VOLUME: `pint.Unit`

Default volume unit for PyAnsys Geometry.

property DefaultUnitsClass.SERVER_LENGTH: `pint.Unit`

Default length unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

property DefaultUnitsClass.SERVER_AREA: `pint.Unit`

Default area unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

property DefaultUnitsClass.SERVER_VOLUME: `pint.Unit`

Default volume unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

property DefaultUnitsClass.SERVER_ANGLE: `pint.Unit`

Default angle unit for gRPC messages.

Notes

The default units on the server side are not modifiable yet.

Measurement

class ansys.geometry.core.misc.measurements.Measurement(*value: ansys.geometry.core.typing.Real | pint.Quantity, unit: pint.Unit, dimensions: pint.Unit*)

Bases: `ansys.geometry.core.misc.units.PhysicalQuantity`

Provides the PhysicalQuantity subclass for holding a measurement.

Parameters

value

[Real | `Quantity`] Value of the measurement.

unit

[`Unit`] Units for the measurement.

dimensions

[`Unit`] Units for extracting the dimensions of the measurement. If `~pint.Unit.meter` is given, the dimension extracted is [`length`].

Overview

Properties

<code>value</code>	Value of the measurement.
--------------------	---------------------------

Special methods

<code>__repr__</code>	Representation of the Measurement class.
<code>__eq__</code>	Equals operator for the Measurement class.

Import detail

```
from ansys.geometry.core.misc.measurements import Measurement
```

Property detail

property `Measurement.value: pint.Quantity`

Value of the measurement.

Method detail

`Measurement.__repr__()`

Representation of the Measurement class.

`Measurement.__eq__(other: Measurement) → bool`

Equals operator for the Measurement class.

Distance

class `ansys.geometry.core.misc.measurements.Distance`(*value: ansys.geometry.core.typing.Real | pint.Quantity, unit: pint.Unit | None = None*)

Bases: Measurement

Provides the Measurement subclass for holding a distance.

Parameters

value

[Real | `Quantity`] Value of the distance.

unit

[`Unit`, default: `DEFAULT_UNITS.LENGTH`] Units for the distance.

Import detail

```
from ansys.geometry.core.misc.measurements import Distance
```

Angle

```
class ansys.geometry.core.misc.measurements.Angle(value: ansys.geometry.core.typing.Real |
                                                    pint.Quantity, unit: pint.Unit | None = None)
```

Bases: Measurement

Provides the Measurement subclass for holding an angle.

Parameters

value

[Real | Quantity] Value of the angle.

unit

[Unit, default: DEFAULT_UNITS.ANGLE] Units for the angle.

Import detail

```
from ansys.geometry.core.misc.measurements import Angle
```

Area

```
class ansys.geometry.core.misc.measurements.Area(value: ansys.geometry.core.typing.Real |
                                                    pint.Quantity, unit: pint.Unit | None = None)
```

Bases: Measurement

Provides the Measurement subclass for holding an area.

Parameters

value

[Real | Quantity] Value of the area.

unit

[Unit, default: DEFAULT_UNITS.AREA] Units for the area.

Import detail

```
from ansys.geometry.core.misc.measurements import Area
```

Description

Provides various measurement-related classes.

Module detail

`ansys.geometry.core.misc.measurements.DEFAULT_UNITS`

PyAnsys Geometry default units object.

The options.py module

Summary

Classes

<i>ImportOptions</i>	Import options when opening a file.
<i>ImportOptionsDefinitions</i>	Import options definitions when opening a file.
<i>TessellationOptions</i>	Provides options for getting tessellation.

ImportOptions

class ansys.geometry.core.misc.options.ImportOptions

Import options when opening a file.

Parameters

cleanup_bodies

[bool = False] Simplify geometry and clean up topology.

import_coordinate_systems

[bool = False] Import coordinate systems.

import_curves

[bool = False] Import curves.

import_hidden_components_and_geometry

[bool = False] Import hidden components and geometry.

import_names

[bool = False] Import names of bodies and curves.

import_planes

[bool = False] Import planes.

import_points

[bool = False] Import points.

import_named_selections

[bool = True] Import the named selections associated with the root component being inserted.

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>
<code>import_named_selections</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`

`ImportOptions.import_named_selections: bool = True`

Method detail

`ImportOptions.to_dict()`

Provide the dictionary representation of the `ImportOptions` class.

`ImportOptionsDefinitions`

class `ansys.geometry.core.misc.options.ImportOptionsDefinitions`

Import options definitions when opening a file.

Parameters

import_named_selections_keys

[`str = None`] Import the named selections keys associated with the root component being inserted.

Overview

Methods

<code>to_dict</code>	Provide the dictionary representation of the <code>ImportOptionsDefinitions</code> class.
----------------------	---

Attributes

```
import_named_selections_keys
```

Import detail

```
from ansys.geometry.core.misc.options import ImportOptionsDefinitions
```

Attribute detail

`ImportOptionsDefinitions.import_named_selections_keys: str = None`

Method detail

`ImportOptionsDefinitions.to_dict()`

Provide the dictionary representation of the `ImportOptionsDefinitions` class.

TessellationOptions

```
class ansys.geometry.core.misc.options.TessellationOptions(surface_deviation: an-
    sys.geometry.core.misc.measurements.Distance
    | pint.Quantity |
    ansys.geometry.core.typing.Real,
    angle_deviation: an-
    sys.geometry.core.misc.measurements.Angle
    | pint.Quantity |
    ansys.geometry.core.typing.Real,
    max_aspect_ratio:
    ansys.geometry.core.typing.Real = 0.0,
    max_edge_length: an-
    sys.geometry.core.misc.measurements.Distance
    | pint.Quantity |
    ansys.geometry.core.typing.Real = 0.0,
    watertight: bool = False)
```

Provides options for getting tessellation.

Parameters

surface_deviation

[Distance | Quantity | Real] The maximum deviation from the true surface position. If a Real is provided, it is assumed to be in the default length unit.

angle_deviation

[Angle | Quantity | Real] The maximum deviation from the true surface normal. If a Real is provided, it is assumed to be in radians.

max_aspect_ratio

[Real, default=0.0] The maximum aspect ratio of facets.

max_edge_length

[Distance | Quantity | Real, default=0.0] The maximum facet edge length.

watertight

[bool, default=False] Whether triangles on opposite sides of an edge should match.

Overview

Properties

<code>surface_deviation</code>	Surface Deviation.
<code>angle_deviation</code>	Angle deviation.
<code>max_aspect_ratio</code>	Maximum aspect ratio.
<code>max_edge_length</code>	Maximum edge length.
<code>watertight</code>	Watertight.

Import detail

```
from ansys.geometry.core.misc.options import TessellationOptions
```

Property detail

property `TessellationOptions.surface_deviation:`

`ansys.geometry.core.misc.measurements.Distance`

Surface Deviation.

The maximum deviation from the true surface position.

property `TessellationOptions.angle_deviation:` `ansys.geometry.core.misc.measurements.Angle`

Angle deviation.

The maximum deviation from the true surface normal.

property `TessellationOptions.max_aspect_ratio:` `ansys.geometry.core.typing.Real`

Maximum aspect ratio.

The maximum aspect ratio of facets.

property `TessellationOptions.max_edge_length:`

`ansys.geometry.core.misc.measurements.Distance`

Maximum edge length.

The maximum facet edge length.

property `TessellationOptions.watertight:` `bool`

Watertight.

Whether triangles on opposite sides of an edge should match.

Description

Provides various option classes.

The `units.py` module

Summary

Classes

<code>PhysicalQuantity</code>	Provides the base class for handling units throughout PyAnsys Geometry.
-------------------------------	---

Constants

<code>UNITS</code>	Units manager.
--------------------	----------------

PhysicalQuantity

class `ansys.geometry.core.misc.units.PhysicalQuantity`(*unit*: `pint.Unit`, *expected_dimensions*: `pint.Unit` | `None` = `None`)

Provides the base class for handling units throughout PyAnsys Geometry.

Parameters

unit

[`Unit`] Units for the class.

expected_dimensions

[`Unit`, default: `None`] Units for the dimensionality of the physical quantity.

Overview

Properties

<code>unit</code>	Unit of the object.
<code>base_unit</code>	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

`ansys.geometry.core.misc.units.UNITS`

Units manager.

Description

Provides the PyAnsys Geometry miscellaneous subpackage.

The parameters package

Summary

Submodules

<i>parameter</i>	Provides get and set methods for parameters.
------------------	--

The parameter.py module

Summary

Classes

<i>Parameter</i>	Represents a parameter.
------------------	-------------------------

Enums

<i>ParameterType</i>	Provides values for the parameter types supported.
<i>ParameterUpdateStatus</i>	Provides values for the status messages associated with parameter updates.

Constants

<i>UNIT_MAP</i>

Parameter

```
class ansys.geometry.core.parameters.parameter.Parameter(id: int, name: str, dimension_type:
                                                         ParameterType, dimension_value:
                                                         pint.Quantity |
                                                         ansys.geometry.core.typing.Real)
```

Represents a parameter.

Parameters

id

[int] Unique ID for the parameter.

name

[str] Name of the parameter.

dimension_type

[ParameterType] Type of the parameter.

dimension_value

[Quantity | Real] Value of the parameter. If a Real, it will be assigned default units based on the dimension_type.

Overview

Properties

<code>name</code>	Get the name of the parameter.
<code>dimension_value</code>	Get the value of the parameter.
<code>dimension_type</code>	Get the type of the parameter.

Attributes

<code>id</code>

Static methods

<code>convert_quantity_to_server_units</code>	Convert a Quantity to a Real value using appropriate units.
---	---

Import detail

```
from ansys.geometry.core.parameters.parameter import Parameter
```

Property detail

property `Parameter.name: str`

Get the name of the parameter.

property `Parameter.dimension_value: pint.Quantity`

Get the value of the parameter.

property `Parameter.dimension_type: ParameterType`

Get the type of the parameter.

Attribute detail

`Parameter.id`

Method detail

static `Parameter.convert_quantity_to_server_units(value: pint.Quantity, dimension_type: ParameterType) → ansys.geometry.core.typing.Real`

Convert a Quantity to a Real value using appropriate units.

Parameters

value

[Quantity] The value to convert.

dimension_type

[ParameterType] The dimension type to determine the appropriate unit.

Returns

Real

The value in default server units.

ParameterType

class ansys.geometry.core.parameters.parameter.**ParameterType**(*args, **kws)

Bases: `enum.Enum`

Provides values for the parameter types supported.

Overview**Attributes**

<i>DIMENSIONTYPE_UNKNOWN</i>
<i>DIMENSIONTYPE_LINEAR</i>
<i>DIMENSIONTYPE_DIAMETRIC</i>
<i>DIMENSIONTYPE_RADIAL</i>
<i>DIMENSIONTYPE_ARC</i>
<i>DIMENSIONTYPE_AREA</i>
<i>DIMENSIONTYPE_VOLUME</i>
<i>DIMENSIONTYPE_MASS</i>
<i>DIMENSIONTYPE_ANGULAR</i>
<i>DIMENSIONTYPE_COUNT</i>
<i>DIMENSIONTYPE_UNITLESS</i>

Import detail

```
from ansys.geometry.core.parameters.parameter import ParameterType
```

Attribute detail

`ParameterType.DIMENSIONTYPE_UNKNOWN = 0`

`ParameterType.DIMENSIONTYPE_LINEAR = 1`

`ParameterType.DIMENSIONTYPE_DIAMETRIC = 2`

`ParameterType.DIMENSIONTYPE_RADIAL = 3`

`ParameterType.DIMENSIONTYPE_ARC = 4`

`ParameterType.DIMENSIONTYPE_AREA = 5`

`ParameterType.DIMENSIONTYPE_VOLUME = 6`

`ParameterType.DIMENSIONTYPE_MASS = 7`

`ParameterType.DIMENSIONTYPE_ANGULAR = 8`

`ParameterType.DIMENSIONTYPE_COUNT = 9`

`ParameterType.DIMENSIONTYPE_UNITLESS = 10`

ParameterUpdateStatus

class ansys.geometry.core.parameters.parameter.ParameterUpdateStatus(*args, **kwargs)

Bases: `enum.Enum`

Provides values for the status messages associated with parameter updates.

Overview

Attributes

<i>SUCCESS</i>
<i>FAILURE</i>
<i>CONSTRAINED_PARAMETERS</i>
<i>UNKNOWN</i>

Import detail

```
from ansys.geometry.core.parameters.parameter import ParameterUpdateStatus
```

Attribute detail

ParameterUpdateStatus.SUCCESS = 0

ParameterUpdateStatus.FAILURE = 1

ParameterUpdateStatus.CONSTRAINED_PARAMETERS = 2

ParameterUpdateStatus.UNKNOWN = 3

Description

Provides get and set methods for parameters.

Module detail

ansys.geometry.core.parameters.parameter.UNIT_MAP

Description

PyAnsys Geometry parameters subpackage.

The plotting package

Summary

Subpackages

<i>widgets</i>	Submodule providing widgets for the PyAnsys Geometry plotter.
----------------	---

Submodules

<code>plotter</code>	Provides plotting for various PyAnsys Geometry objects.
----------------------	---

The widgets package

Summary

Submodules

<code>show_design_point</code>	Provides the ruler widget for the PyAnsys Geometry plotter.
--------------------------------	---

The `show_design_point.py` module

Summary

Classes

<code>ShowDesignPoints</code>	Provides the a button to hide/show DesignPoint objects in the plotter.
-------------------------------	--

ShowDesignPoints

class `ansys.geometry.core.plotting.widgets.show_design_point.ShowDesignPoints`(*plotter_helper*:

an-

sys.tools.visualization_interface.l

Bases: `ansys.tools.visualization_interface.backends.pyvista.widgets.PlotterWidget`

Provides the a button to hide/show DesignPoint objects in the plotter.

Parameters

plotter_helper

[GeometryPlotter] Provides the plotter to add the button to.

Overview

Methods

<code>callback</code>	Remove or add the DesignPoint actors upon click.
<code>update</code>	Define the configuration and representation of the button widget.

Attributes

<code>plotter_helper</code>

Import detail

```
from ansys.geometry.core.plotting.widgets.show_design_point import ShowDesignPoints
```

Attribute detail

ShowDesignPoints.**plotter_helper**

Method detail

ShowDesignPoints.**callback**(*state: bool*) → None

Remove or add the DesignPoint actors upon click.

Parameters

state

[bool] State of the button, which is inherited from PyVista. The value is True if the button is active.

ShowDesignPoints.**update**() → None

Define the configuration and representation of the button widget.

Description

Provides the ruler widget for the PyAnsys Geometry plotter.

Description

Submodule providing widgets for the PyAnsys Geometry plotter.

The plotter.py module

Summary

Classes

<i>GeometryPlotter</i> Plotter for PyAnsys Geometry objects.
--

Constants

<i>POLYDATA_COLOR_CYCLER</i>

GeometryPlotter

```
class ansys.geometry.core.plotting.plotter.GeometryPlotter(use_trame: bool | None = None,  
                                                         use_service_colors: bool | None =  
                                                         None, allow_picking: bool = False,  
                                                         show_plane: bool = True)
```

Bases: ansys.tools.visualization_interface.Plotter

Plotter for PyAnsys Geometry objects.

This class is an implementation of the PlotterInterface class.

Parameters

use_trame

[bool, optional] Whether to use trame visualizer or not, by default None.

use_service_colors

[bool, optional] Whether to use service colors or not, by default None.

allow_picking

[bool, optional] Whether to allow picking or not, by default False.

show_plane

[bool, optional] Whether to show the plane in the scene, by default True.

Overview**Methods**

<code>add_frame</code>	Plot a frame in the scene.
<code>add_plane</code>	Plot a plane in the scene.
<code>add_sketch</code>	Plot a sketch in the scene.
<code>add_surface</code>	Add a surface to the scene.
<code>add_body_edges</code>	Add the outer edges of a body to the plot.
<code>add_body</code>	Add a body to the scene.
<code>add_face</code>	Add a face to the scene.
<code>add_component</code>	Add a component to the scene.
<code>add_component_by_body</code>	Add a component on a per body basis.
<code>add_sketch_polydata</code>	Add sketches to the scene from PyVista polydata.
<code>add_design_point</code>	Add a DesignPoint object to the plotter.
<code>plot_iter</code>	Add a list of any type of object to the scene.
<code>plot</code>	Add a custom mesh to the plotter.
<code>show</code>	Show the plotter.
<code>export_glb</code>	Export the design to a glb file. Does not support picked objects.

Properties

<code>use_service_colors</code>	Indicates whether to use service colors for plotting purposes.
---------------------------------	--

Import detail

```
from ansys.geometry.core.plotting.plotter import GeometryPlotter
```

Property detail

property `GeometryPlotter.use_service_colors`: **bool**

Indicates whether to use service colors for plotting purposes.

Method detail

`GeometryPlotter.add_frame`(*frame*: `ansys.geometry.core.math.frame.Frame`, *plotting_options*: `dict` | `None` = `None`) → `None`

Plot a frame in the scene.

Parameters**frame**

[Frame] Frame to render in the scene.

plotting_options

[dict, default: None] dictionary containing parameters accepted by the `pyvista.create_axes_marker()` class for customizing the frame rendering in the scene.

`GeometryPlotter.add_plane(plane: ansys.geometry.core.math.plane.Plane, plane_options: dict | None = None, plotting_options: dict | None = None) → None`

Plot a plane in the scene.

Parameters

plane

[Plane] Plane to render in the scene.

plane_options

[dict, default: None] dictionary containing parameters accepted by the `pyvista.Plane` function for customizing the mesh representing the plane.

plotting_options

[dict, default: None] dictionary containing parameters accepted by the `Plotter.add_mesh` method for customizing the mesh rendering of the plane.

`GeometryPlotter.add_sketch(sketch: ansys.geometry.core.sketch.sketch.Sketch, show_plane: bool = False, show_frame: bool = False, **plotting_options: dict | None) → None`

Plot a sketch in the scene.

Parameters

sketch

[Sketch] Sketch to render in the scene.

show_plane

[bool, default: False] Whether to render the sketch plane in the scene.

show_frame

[bool, default: False] Whether to show the frame in the scene.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_surface(surface: ansys.geometry.core.shapes-surfaces.Surface, **plotting_options) → None`

Add a surface to the scene.

Parameters

surface

[Surface] Surface to add.

****plotting_options**

[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_body_edges(body_plot: ansys.tools.visualization-interface.MeshObjectPlot, **plotting_options: dict | None) → None`

Add the outer edges of a body to the plot.

This method has the side effect of adding the edges to the `GeomObject` that you pass through the parameters.

Parameters

body_plot
[MeshObjectPlot] Body of which to add the edges.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_body`(*body*: `ansys.geometry.core.designer.body.Body`, *merge*: *bool* = *False*,
***plotting_options*: *dict* | *None*) → *None*

Add a body to the scene.

Parameters

body
[Body] Body to add.

merge
[bool, default: *False*] Whether to merge the body into a single mesh. When *True*, the individual faces of the tessellation are merged. This preserves the number of triangles and only merges the topology.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_face`(*face*: `ansys.geometry.core.designer.face.Face`, ***plotting_options*: *dict* | *None*) → *None*

Add a face to the scene.

Parameters

face
[Face] Face to add.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_component`(*component*: `ansys.geometry.core.designer.component.Component`,
merge_component: *bool* = *False*, *merge_bodies*: *bool* = *False*,
***plotting_options*) → *None*

Add a component to the scene.

Parameters

component
[Component] Component to add.

merge_component
[bool, default: *False*] Whether to merge the component into a single dataset. When *True*, all the individual bodies are effectively combined into a single dataset without any hierarchy.

merge_bodies
[bool, default: *False*] Whether to merge each body into a single dataset. When *True*, all the faces of each individual body are effectively combined into a single dataset without separating faces.

****plotting_options**
[dict, default: None] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_component_by_body`(*component*: `ansys.geometry.core.designer.component.Component`,
merge_bodies: `bool`, ***plotting_options*: `dict | None`) → `None`

Add a component on a per body basis.

Parameters

component

[`Component`] Component to add.

****plotting_options**

[`dict`, default: `None`] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

Notes

This will allow to make use of the service colors. At the same time, it will be slower than the `add_component` method.

`GeometryPlotter.add_sketch_polydata`(*polydata_entries*: `list[pyvista.PolyData]`, *sketch*:
`ansys.geometry.core.sketch.sketch.Sketch` = `None`,
***plotting_options*) → `None`

Add sketches to the scene from PyVista polydata.

Parameters

polydata_entries

[`list[pyvista.PolyData]`] Polydata to add.

sketch

[`Sketch`, default: `None`] Sketch to add.

****plotting_options**

[`dict`, default: `None`] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.add_design_point`(*design_point*: `ansys.geometry.core.designer.designpoint.DesignPoint`,
***plotting_options*) → `None`

Add a `DesignPoint` object to the plotter.

Parameters

design_point

[`DesignPoint`] `DesignPoint` to add.

`GeometryPlotter.plot_iter`(*plotting_list*: `list[Any]`, *name_filter*: `str` = `None`, ***plotting_options*) → `None`

Add a list of any type of object to the scene.

These types of objects are supported: `Body`, `Component`, `list[pv.PolyData]`, `pv.MultiBlock`, and `Sketch`.

Parameters

plotting_list

[`list[Any]`] list of objects you want to plot.

name_filter

[`str`, default: `None`] Regular expression with the desired name or names you want to include in the plotter.

****plotting_options**

[`dict`, default: `None`] Keyword arguments. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`GeometryPlotter.plot(plottable_object: Any, name_filter: str = None, **plotting_options) → None`

Add a custom mesh to the plotter.

Parameters

plottable_object

[`str`, default: `None`] Regular expression with the desired name or names you want to include in the plotter.

name_filter: str, default: None

Regular expression with the desired name or names you want to include in the plotter.

****plotting_options**

[`dict`, default: `None`] Keyword arguments. For allowable keyword arguments, depend of the backend implementation you are using.

`GeometryPlotter.show(plotting_object: Any = None, screenshot: str | None = None, **plotting_options) → None | list[Any]`

Show the plotter.

Parameters

plotting_object

[`Any`, default: `None`] Object you can add to the plotter.

screenshot

[`str`, default: `None`] Path to save a screenshot of the plotter.

****plotting_options**

[`dict`, default: `None`] Keyword arguments for the plotter. Arguments depend of the backend implementation you are using.

`GeometryPlotter.export_glb(plotting_object: Any = None, filename: str | pathlib.Path | None = None, **plotting_options) → pathlib.Path`

Export the design to a glb file. Does not support picked objects.

Parameters

plotting_object

[`Any`, default: `None`] Object you can add to the plotter.

filename

[`str` | `Path`, default: `None`] Path to save a GLB file of the plotter. If `None`, the file will be saved as `temp_glb.glb`.

****plotting_options**

[`dict`, default: `None`] Keyword arguments for the plotter. Arguments depend of the backend implementation you are using.

Returns

Path

Path to the exported glb file.

Description

Provides plotting for various PyAnsys Geometry objects.

Module detail

`ansys.geometry.core.plotting.plotter.POLYDATA_COLOR_CYCLER`

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>nurbs</i>	Provides for creating and managing a NURBS curve.
<i>trimmed_curve</i>	Trimmed curve class.

The circle.py module

Summary

Classes

<i>Circle</i>	Provides 3D circle representation.
<i>CircleEvaluation</i>	Provides evaluation of a circle at a given parameter.

Circle

```

class ansys.geometry.core.shapes.curves.circle.Circle(origin: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.point.Point3D,
radius: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real, reference:
numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
an-
sys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_X, axis: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
an-
sys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_Z)

```

Bases: `ansys.geometry.core.shapes.curves.curve.Curve`

Provides 3D circle representation.

Parameters

origin

[`ndarray` | `RealSequence` | `Point3D`] Origin of the circle.

radius

[`Quantity` | `Distance` | `Real`] Radius of the circle.

reference

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] X-axis direction.

axis

[`ndarray` | `RealSequence` | `UnitVector3D` | `Vector3D`] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the curve.
<code>contains_point</code>	Check a point is contained by the curve.

Methods

<code>evaluate</code>	Evaluate the circle at a given parameter.
<code>transformed_copy</code>	Create a transformed copy of the circle from 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

<i>origin</i>	Origin of the circle.
<i>radius</i>	Radius of the circle.
<i>diameter</i>	Diameter of the circle.
<i>perimeter</i>	Perimeter of the circle.
<i>area</i>	Area of the circle.
<i>dir_x</i>	X-direction of the circle.
<i>dir_y</i>	Y-direction of the circle.
<i>dir_z</i>	Z-direction of the circle.

Special methods

<code>__eq__</code>	Equals operator for the Circle class.
---------------------	---------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.circle import Circle
```

Property detail

property Circle.*origin*: *ansys.geometry.core.math.point.Point3D*

Origin of the circle.

property Circle.*radius*: *pint.Quantity*

Radius of the circle.

property Circle.*diameter*: *pint.Quantity*

Diameter of the circle.

property Circle.*perimeter*: *pint.Quantity*

Perimeter of the circle.

property Circle.*area*: *pint.Quantity*

Area of the circle.

property Circle.*dir_x*: *ansys.geometry.core.math.vector.UnitVector3D*

X-direction of the circle.

property Circle.*dir_y*: *ansys.geometry.core.math.vector.UnitVector3D*

Y-direction of the circle.

property Circle.*dir_z*: *ansys.geometry.core.math.vector.UnitVector3D*

Z-direction of the circle.

Method detail

Circle.*__eq__*(*other*: Circle) → bool

Equals operator for the Circle class.

Circle.*evaluate*(*parameter*: *ansys.geometry.core.typing.Real*) → *CircleEvaluation*

Evaluate the circle at a given parameter.

Parameters**parameter**

[Real] Parameter to evaluate the circle at.

Returns**CircleEvaluation**

Resulting evaluation.

Circle.transformed_copy(*matrix*: [ansys.geometry.core.math.matrix.Matrix44](#)) → [Circle](#)

Create a transformed copy of the circle from a transformation matrix.

Parameters**matrix**

[Matrix44] 4x4 transformation matrix to apply to the circle.

Returns**Circle**

New circle that is the transformed copy of the original circle.

Circle.mirrored_copy() → [Circle](#)

Create a mirrored copy of the circle along the y-axis.

Returns**Circle**

A new circle that is a mirrored copy of the original circle.

Circle.project_point(*point*: [ansys.geometry.core.math.point.Point3D](#)) → [CircleEvaluation](#)

Project a point onto the circle and evaluate the circle.

Parameters**point**

[Point3D] Point to project onto the circle.

Returns**CircleEvaluation**

Resulting evaluation.

Circle.is_coincident_circle(*other*: [Circle](#)) → [bool](#)

Determine if the circle is coincident with another.

Parameters**other**

[Circle] Circle to determine coincidence with.

Returns**bool**

True if this circle is coincident with the other, False otherwise.

Circle.parameterization() → [ansys.geometry.core.shapes.parameterization.Parameterization](#)

Get the parametrization of the circle.

The parameter of a circle specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

Returns

Parameterization

Information about how the circle is parameterized.

abstractmethod `Circle.contains_param(param: ansys.geometry.core.typing.Real) → bool`

Check a parameter is within the parametric range of the curve.

abstractmethod `Circle.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

CircleEvaluation

class `ansys.geometry.core.shapes.curves.circle.CircleEvaluation(circle: Circle, parameter: ansys.geometry.core.typing.Real)`

Bases: `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Provides evaluation of a circle at a given parameter.

Parameters

circle: `~ansys.geometry.core.shapes.curves.circle.Circle`

Circle to evaluate.

parameter: `Real`

Parameter to evaluate the circle at.

Overview

Properties

<code>circle</code>	Circle being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>tangent</code>	Tangent of the evaluation.
<code>normal</code>	Normal to the circle.
<code>first_derivative</code>	First derivative of the evaluation.
<code>second_derivative</code>	Second derivative of the evaluation.
<code>curvature</code>	Curvature of the circle.

Import detail

```
from ansys.geometry.core.shapes.curves.circle import CircleEvaluation
```

Property detail

property `CircleEvaluation.circle: Circle`

Circle being evaluated.

property `CircleEvaluation.parameter: ansys.geometry.core.typing.Real`

Parameter that the evaluation is based upon.

property `CircleEvaluation.position: ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

Point3D

Point that lies on the circle at this evaluation.

property CircleEvaluation.tangent: *ansys.geometry.core.math.vector.UnitVector3D*

Tangent of the evaluation.

Returns**UnitVector3D**

Tangent unit vector to the circle at this evaluation.

property CircleEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the circle.

Returns**UnitVector3D**

Normal unit vector to the circle at this evaluation.

property CircleEvaluation.first_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative of the evaluation.

The first derivative is in the direction of the tangent and has a magnitude equal to the velocity (rate of change of position) at that point.

Returns**Vector3D**

First derivative of the evaluation.

property CircleEvaluation.second_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative of the evaluation.

Returns**Vector3D**

Second derivative of the evaluation.

property CircleEvaluation.curvature: *ansys.geometry.core.typing.Real*

Curvature of the circle.

Returns**Real**

Curvature of the circle.

Description

Provides for creating and managing a circle.

The curve.py module**Summary****Classes**

<i>Curve</i>	Provides the abstract base class representing a 3D curve.
--------------	---

Curve

class `ansys.geometry.core.shapes.curves.curve.Curve`

Bases: `abc.ABC`

Provides the abstract base class representing a 3D curve.

Overview

Abstract methods

<code>parameterization</code>	Parameterize the curve.
<code>contains_param</code>	Check a parameter is within the parametric range of the curve.
<code>contains_point</code>	Check a point is contained by the curve.
<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

abstractmethod `Curve.parameterization()` → *`ansys.geometry.core.shapes.parameterization.Parameterization`*
Parameterize the curve.

abstractmethod `Curve.contains_param(param: ansys.geometry.core.typing.Real)` → `bool`
Check a parameter is within the parametric range of the curve.

abstractmethod `Curve.contains_point(point: ansys.geometry.core.math.point.Point3D)` → `bool`
Check a point is contained by the curve.
The point can either lie within the curve or on its boundary.

abstractmethod `Curve.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44)` → *`Curve`*
Create a transformed copy of the curve.

abstractmethod `Curve.__eq__(other: Curve)` → `bool`
Determine if two curves are equal.

abstractmethod `Curve.evaluate(parameter: ansys.geometry.core.typing.Real)` → *`ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`*
Evaluate the curve at the given parameter.

abstractmethod `Curve.project_point(point: ansys.geometry.core.math.point.Point3D) → ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Project a point to the curve.

This method returns the evaluation at the closest point.

`Curve.trim(interval: ansys.geometry.core.shapes.parameterization.Interval) → ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve`

Trim this curve by bounding it with an interval.

Returns

TrimmedCurve

The resulting bounded curve.

Description

Provides the Curve class.

The curve_evaluation.py module

Summary

Classes

<code>CurveEvaluation</code>	Provides for evaluating a curve.
------------------------------	----------------------------------

CurveEvaluation

class `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation(parameter: ansys.geometry.core.typing.Real = None)`

Provides for evaluating a curve.

Overview

Methods

<code>is_set</code>	Determine if the parameter for the evaluation has been set.
---------------------	---

Properties

<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>first_derivative</code>	First derivative of the evaluation.
<code>second_derivative</code>	Second derivative of the evaluation.
<code>curvature</code>	Curvature of the evaluation.

Import detail

```
from ansys.geometry.core.shapes.curves.curve_evaluation import CurveEvaluation
```

Property detail

property CurveEvaluation.parameter: *ansys.geometry.core.typing.Real*

Abstractmethod

Parameter that the evaluation is based upon.

property CurveEvaluation.position: *ansys.geometry.core.math.point.Point3D*

Abstractmethod

Position of the evaluation.

property CurveEvaluation.first_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

First derivative of the evaluation.

property CurveEvaluation.second_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

Second derivative of the evaluation.

property CurveEvaluation.curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Curvature of the evaluation.

Method detail

CurveEvaluation.is_set() → bool

Determine if the parameter for the evaluation has been set.

Description

Provides for creating and managing a curve.

The ellipse.py module

Summary

Classes

<i>Ellipse</i>	Provides 3D ellipse representation.
<i>EllipseEvaluation</i>	Evaluate an ellipse at a given parameter.

Ellipse

```
class ansys.geometry.core.shapes.curves.ellipse.Ellipse(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D,
    major_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real,
    minor_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real,
    reference: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D
    |
    ansys.geometry.core.math.vector.Vector3D
    = UNITVECTOR3D_X, axis:
    numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D
    |
    ansys.geometry.core.math.vector.Vector3D
    = UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes.curves.curve.Curve*

Provides 3D ellipse representation.

Parameters

origin

[*ndarray* | RealSequence | Point3D] Origin of the ellipse.

major_radius

[*Quantity* | Distance | Real] Major radius of the ellipse.

minor_radius

[*Quantity* | Distance | Real] Minor radius of the ellipse.

reference

[*ndarray* | RealSequence | UnitVector3D | Vector3D] X-axis direction.

axis

[*ndarray* | RealSequence | UnitVector3D | Vector3D] Z-axis direction.

Overview

Abstract methods

<i>contains_param</i>	Check a parameter is within the parametric range of the curve.
<i>contains_point</i>	Check a point is contained by the curve.

Methods

<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 from 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 from a transformation matrix.

Parameters

matrix

[Matrix44] 4x4 transformation matrix to apply to the ellipse.

Returns

Ellipse

New ellipse that is the transformed copy of the original ellipse.

`Ellipse.parameterization() → ansys.geometry.core.shapes.parameterization.Parameterization`

Get the parametrization of the ellipse.

The parameter of an ellipse specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

Returns

Parameterization

Information about how the ellipse is parameterized.

abstractmethod `Ellipse.contains_param(param: ansys.geometry.core.typing.Real) → bool`

Check a parameter is within the parametric range of the curve.

abstractmethod `Ellipse.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

EllipseEvaluation

class `ansys.geometry.core.shapes.curves.ellipse.EllipseEvaluation(ellipse: Ellipse, parameter: ansys.geometry.core.typing.Real)`

Bases: `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate an ellipse at a given parameter.

Parameters

ellipse: ~ansys.geometry.core.shapes.curves.ellipse.Ellipse

Ellipse to evaluate.

parameter: float, int

Parameter to evaluate the ellipse at.

Overview

Properties

<i>ellipse</i>	Ellipse being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>tangent</i>	Tangent of the evaluation.
<i>normal</i>	Normal of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the ellipse.

Import detail

```
from ansys.geometry.core.shapes.curves.ellipse import EllipseEvaluation
```

Property detail

property EllipseEvaluation.ellipse: *Ellipse*

Ellipse being evaluated.

property EllipseEvaluation.parameter: *ansys.geometry.core.typing.Real*

Parameter that the evaluation is based upon.

property EllipseEvaluation.position: *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the ellipse at this evaluation.

property EllipseEvaluation.tangent: *ansys.geometry.core.math.vector.UnitVector3D*

Tangent of the evaluation.

Returns

UnitVector3D

Tangent unit vector to the ellipse at this evaluation.

property EllipseEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

Normal of the evaluation.

Returns

UnitVector3D

Normal unit vector to the ellipse at this evaluation.

property EllipseEvaluation.first_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative of the evaluation.

The first derivative is in the direction of the tangent and has a magnitude equal to the velocity (rate of change of position) at that point.

Returns

Vector3D

First derivative of the evaluation.

property `EllipseEvaluation.second_derivative`: [ansys.geometry.core.math.vector.Vector3D](#)

Second derivative of the evaluation.

Returns

Vector3D

Second derivative of the evaluation.

property `EllipseEvaluation.curvature`: [ansys.geometry.core.typing.Real](#)

Curvature of the ellipse.

Returns

Real

Curvature of the ellipse.

Description

Provides for creating and managing an ellipse.

The `line.py` module

Summary

Classes

Line	Provides 3D line representation.
LineEvaluation	Provides for evaluating a line.

Line

class `ansys.geometry.core.shapes.curves.line.Line`(*origin*: [numpy.ndarray](#) | [ansys.geometry.core.typing.RealSequence](#) | [ansys.geometry.core.math.point.Point3D](#), *direction*: [numpy.ndarray](#) | [ansys.geometry.core.typing.RealSequence](#) | [ansys.geometry.core.math.vector.UnitVector3D](#) | [ansys.geometry.core.math.vector.Vector3D](#))

Bases: [ansys.geometry.core.shapes.curves.curve.Curve](#)

Provides 3D line representation.

Parameters

origin

[[ndarray](#) | [RealSequence](#) | [Point3D](#)] Origin of the line.

direction

[[ndarray](#) | [RealSequence](#) | [UnitVector3D](#) | [Vector3D](#)] Direction of the line.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the curve.
<code>contains_point</code>	Check a point is contained by the curve.

Methods

<code>evaluate</code>	Evaluate the line at a given parameter.
<code>transformed_copy</code>	Create a transformed copy of the line from a transformation matrix.
<code>project_point</code>	Project a point onto the line and evaluate the line.
<code>is_coincident_line</code>	Determine if the line is coincident with another line.
<code>is_opposite_line</code>	Determine if the line is opposite another line.
<code>parameterization</code>	Get the parametrization of the line.

Properties

<code>origin</code>	Origin of the line.
<code>direction</code>	Direction of the line.

Special methods

<code>__eq__</code>	Equals operator for the Line class.
---------------------	-------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.line import Line
```

Property detail

property `Line.origin`: `ansys.geometry.core.math.point.Point3D`

Origin of the line.

property `Line.direction`: `ansys.geometry.core.math.vector.UnitVector3D`

Direction of the line.

Method detail

`Line.__eq__(other: object)` → bool

Equals operator for the Line class.

`Line.evaluate(parameter: float)` → `LineEvaluation`

Evaluate the line at a given parameter.

Parameters

parameter

[Real] Parameter to evaluate the line at.

Returns**LineEvaluation**

Resulting evaluation.

`Line.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Line`

Create a transformed copy of the line from a transformation matrix.

Parameters**matrix**

[Matrix44] 4X4 transformation matrix to apply to the line.

Returns**Line**

New line that is the transformed copy of the original line.

`Line.project_point(point: ansys.geometry.core.math.point.Point3D) → LineEvaluation`

Project a point onto the line and evaluate the line.

Parameters**point**

[Point3D] Point to project onto the line.

Returns**LineEvaluation**

Resulting evaluation.

`Line.is_coincident_line(other: Line) → bool`

Determine if the line is coincident with another line.

Parameters**other**

[Line] Line to determine coincidence with.

Returns**bool**

True if the line is coincident with another line, False otherwise.

`Line.is_opposite_line(other: Line) → bool`

Determine if the line is opposite another line.

Parameters**other**

[Line] Line to determine opposition with.

Returns**bool**

True if the line is opposite to another line.

`Line.parameterization() → ansys.geometry.core.shapes.parameterization.Parameterization`

Get the parametrization of the line.

The parameter of a line specifies the distance from the *origin* in the direction of *direction*.

Returns

Parameterization

Information about how the line is parameterized.

abstractmethod `Line.contains_param(param: ansys.geometry.core.typing.Real) → bool`

Check a parameter is within the parametric range of the curve.

abstractmethod `Line.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

LineEvaluation

class `ansys.geometry.core.shapes.curves.line.LineEvaluation(line: Line, parameter: float = None)`

Bases: `ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Provides for evaluating a line.

Overview

Properties

<code>line</code>	Line being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>tangent</code>	Tangent of the evaluation.
<code>first_derivative</code>	First derivative of the evaluation.
<code>second_derivative</code>	Second derivative of the evaluation.
<code>curvature</code>	Curvature of the line, which is always 0.

Import detail

```
from ansys.geometry.core.shapes.curves.line import LineEvaluation
```

Property detail

property `LineEvaluation.line: Line`

Line being evaluated.

property `LineEvaluation.parameter: float`

Parameter that the evaluation is based upon.

property `LineEvaluation.position: ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

Point3D

Point that lies on the line at this evaluation.

property `LineEvaluation.tangent: ansys.geometry.core.math.vector.UnitVector3D`

Tangent of the evaluation.

Returns

UnitVector3D

Tangent unit vector to the line at this evaluation.

Notes

This is always equal to the direction of the line.

property `LineEvaluation.first_derivative`: `ansys.geometry.core.math.vector.Vector3D`

First derivative of the evaluation.

The first derivative is always equal to the direction of the line.

Returns

Vector3D

First derivative of the evaluation.

property `LineEvaluation.second_derivative`: `ansys.geometry.core.math.vector.Vector3D`

Second derivative of the evaluation.

The second derivative is always equal to a zero vector `Vector3D([0, 0, 0])`.

Returns

Vector3D

Second derivative of the evaluation, which is always `Vector3D([0, 0, 0])`.

property `LineEvaluation.curvature`: `float`

Curvature of the line, which is always 0.

Returns

Real

Curvature of the line, which is always 0.

Description

Provides for creating and managing a line.

The `nurbs.py` module

Summary

Classes

<code>NURBSCurve</code>	Represents a NURBS curve.
<code>NURBSCurveEvaluation</code>	Provides evaluation of a NURBS curve at a given parameter.

NURBSCurve

class `ansys.geometry.core.shapes.curves.nurbs.NURBSCurve`(*geomdl_object*: `geomdl.NURBS.Curve` = `None`)

Bases: `ansys.geometry.core.shapes.curves.curve.Curve`

Represents a NURBS curve.

Notes

This class is a wrapper around the NURBS curve class from the *geomdl* library. By leveraging the *geomdl* library, this class provides a high-level interface to create and manipulate NURBS curves. The *geomdl* library is a powerful library for working with NURBS curves and surfaces. For more information, see <https://pypi.org/project/geomdl/>.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the curve.
<code>contains_point</code>	Check a point is contained by the curve.

Constructors

<code>from_control_points</code>	Create a NURBS curve from control points.
<code>fit_curve_from_points</code>	Fit a NURBS curve to a set of points.

Methods

<code>length</code>	Calculate the length of the NURBS curve.
<code>parameterization</code>	Get the parametrization of the NURBS curve.
<code>transformed_copy</code>	Create a transformed copy of the curve.
<code>evaluate</code>	Evaluate the curve at the given parameter.
<code>project_point</code>	Project a point to the NURBS curve.

Properties

<code>geomdl_nurbs_curve</code>	Get the underlying NURBS curve.
<code>control_points</code>	Get the control points of the curve.
<code>degree</code>	Get the degree of the curve.
<code>knots</code>	Get the knot vector of the curve.
<code>weights</code>	Get the weights of the control points.

Special methods

<code>__eq__</code>	Determine if two curves are equal.
---------------------	------------------------------------

Import detail

```
from ansys.geometry.core.shapes.curves.nurbs import NURBSCurve
```

Property detail

property `NURBSCurve.geomdl_nurbs_curve`: `geomdl.NURBS.Curve`

Get the underlying NURBS curve.

Notes

This property gives access to the full functionality of the NURBS curve coming from the `geomdl` library. Use with caution.

property `NURBSCurve.control_points`: `list[ansys.geometry.core.math.Point3D]`

Get the control points of the curve.

property `NURBSCurve.degree: int`

Get the degree of the curve.

property `NURBSCurve.knots: list[ansys.geometry.core.typing.Real]`

Get the knot vector of the curve.

property `NURBSCurve.weights: list[ansys.geometry.core.typing.Real]`

Get the weights of the control points.

Method detail

`NURBSCurve.length(num_points: int = None) → ansys.geometry.core.typing.Real`

Calculate the length of the NURBS curve.

Parameters

num_points

[int, default: None] Number of points to sample along the curve for length calculation.

Returns

Real

Length of the NURBS curve.

classmethod `NURBSCurve.from_control_points(control_points: list[ansys.geometry.core.math.Point3D], degree: int, knots: list[ansys.geometry.core.typing.Real], weights: list[ansys.geometry.core.typing.Real] = None) → NURBSCurve`

Create a NURBS curve from control points.

Parameters

control_points

[list[Point3D]] Control points of the curve.

degree

[int] Degree of the curve.

knots

[list[Real]] Knot vector of the curve.

weights

[list[Real], optional] Weights of the control points.

Returns

NURBSCurve

NURBS curve.

classmethod `NURBSCurve.fit_curve_from_points(points: list[ansys.geometry.core.math.Point3D], degree: int) → NURBSCurve`

Fit a NURBS curve to a set of points.

Parameters

points

[list[Point3D]] Points to fit the curve to.

degree

[int] Degree of the curve.

Returns

NURBSCurve

Fitted NURBS curve.

NURBSCurve.__eq__(*other*: [NURBSCurve](#)) → bool

Determine if two curves are equal.

NURBSCurve.parameterization() → [ansys.geometry.core.shapes.parameterization.Parameterization](#)

Get the parametrization of the NURBS curve.

The parameter is defined in the interval [0, 1] by default. Information is provided about the parameter type and form.

Returns**Parameterization**

Information about how the NURBS curve is parameterized.

NURBSCurve.transformed_copy(*matrix*: [ansys.geometry.core.math.Matrix44](#)) → [NURBSCurve](#)

Create a transformed copy of the curve.

Parameters**matrix**

[Matrix44] Transformation matrix.

Returns**NURBSCurve**

Transformed copy of the curve.

NURBSCurve.evaluate(*parameter*: [ansys.geometry.core.typing.Real](#)) → [ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation](#)

Evaluate the curve at the given parameter.

Parameters**parameter**

[Real] Parameter to evaluate the curve at.

Returns**CurveEvaluation**

Evaluation of the curve at the given parameter.

abstractmethod NURBSCurve.contains_param(*param*: [ansys.geometry.core.typing.Real](#)) → bool

Check a parameter is within the parametric range of the curve.

abstractmethod NURBSCurve.contains_point(*point*: [ansys.geometry.core.math.Point3D](#)) → bool

Check a point is contained by the curve.

The point can either lie within the curve or on its boundary.

NURBSCurve.project_point(*point*: [ansys.geometry.core.math.Point3D](#), *initial_guess*: [ansys.geometry.core.typing.Real](#) | *None* = *None*) → [ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation](#)

Project a point to the NURBS curve.

This method returns the evaluation at the closest point.

Parameters**point**

[Point3D] Point to project to the curve.

initial_guess

[Real, optional] Initial guess for the optimization algorithm. If not provided, the midpoint of the domain is used.

Returns**CurveEvaluation**

Evaluation at the closest point on the curve.

Notes

Based on [the NURBS book](#), the projection of a point to a NURBS curve is the solution to the following optimization problem: minimize the distance between the point and the curve. The distance is defined as the Euclidean distance squared. For more information, please refer to the implementation of the *distance_squared* function.

NURBSCurveEvaluation

```
class ansys.geometry.core.shapes.curves.nurbs.NURBSCurveEvaluation(nurbs_curve: NURBSCurve,  
                                                                    parameter: an-  
                                                                    sys.geometry.core.typing.Real)
```

Bases: *ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation*

Provides evaluation of a NURBS curve at a given parameter.

Parameters

nurbs_curve: ~*ansys.geometry.core.shapes.curves.nurbs.NURBSCurve*
NURBS curve to evaluate.

parameter: Real
Parameter to evaluate the NURBS curve at.

Overview**Properties**

<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>first_derivative</i>	First derivative of the evaluation.
<i>second_derivative</i>	Second derivative of the evaluation.
<i>curvature</i>	Curvature of the evaluation.

Import detail

```
from ansys.geometry.core.shapes.curves.nurbs import NURBSCurveEvaluation
```

Property detail

property NURBSCurveEvaluation.**parameter:** *ansys.geometry.core.typing.Real*
Parameter that the evaluation is based upon.

property NURBSCurveEvaluation.**position:** *ansys.geometry.core.math.Point3D*
Position of the evaluation.

property `NURBSCurveEvaluation.first_derivative`: `ansys.geometry.core.math.vector.Vector3D`

First derivative of the evaluation.

property `NURBSCurveEvaluation.second_derivative`: `ansys.geometry.core.math.vector.Vector3D`

Second derivative of the evaluation.

property `NURBSCurveEvaluation.curvature`: `ansys.geometry.core.typing.Real`

Curvature of the evaluation.

Description

Provides for creating and managing a NURBS curve.

The `trimmed_curve.py` module

Summary

Classes

<code>TrimmedCurve</code>	Represents a trimmed curve.
<code>ReversedTrimmedCurve</code>	Represents a reversed trimmed curve.

`TrimmedCurve`

```
class ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve(geometry: an-
                                                                    sys.geometry.core.shapes.curves.curve.Curve,
                                                                    start: an-
                                                                    sys.geometry.core.math.point.Point3D,
                                                                    end: an-
                                                                    sys.geometry.core.math.point.Point3D,
                                                                    interval: an-
                                                                    sys.geometry.core.shapes.parameterization.Interval,
                                                                    length: pint.Quantity,
                                                                    grpc_client: an-
                                                                    sys.geometry.core.connection.client.GrpcClient
                                                                    = None)
```

Represents a trimmed curve.

A trimmed curve is a curve that has a boundary. This boundary comes in the form of an interval.

Parameters

geometry
[Curve] Underlying mathematical representation of the curve.

start
[Point3D] Start point of the curve.

end
[Point3D] End point of the curve.

interval
[Interval] Parametric interval that bounds the curve.

length
[Quantity] Length of the curve.

grpc_client

[GrpcClient, default: `None`] gRPC client that is required for advanced functions such as `intersect_curve()`.

Overview**Methods**

<code>evaluate_proportion</code>	Evaluate the curve at a proportional value.
<code>intersect_curve</code>	Get the intersect points of this trimmed curve with another one.
<code>transformed_copy</code>	Return a copy of the trimmed curve transformed by the given matrix.
<code>translate</code>	Translate the trimmed curve by a given vector and distance.
<code>rotate</code>	Rotate the trimmed curve around a given axis centered at a given point.

Properties

<code>geometry</code>	Underlying mathematical curve.
<code>start</code>	Start point of the curve.
<code>end</code>	End point of the curve.
<code>length</code>	Calculated length of the edge.
<code>interval</code>	Interval of the curve that provides its boundary.

Special methods

<code>__repr__</code>	Represent the trimmed curve as a string.
-----------------------	--

Import detail

```
from ansys.geometry.core.shapes.curves.trimmed_curve import TrimmedCurve
```

Property detail

property `TrimmedCurve.geometry`: `ansys.geometry.core.shapes.curves.curve.Curve`

Underlying mathematical curve.

property `TrimmedCurve.start`: `ansys.geometry.core.math.point.Point3D`

Start point of the curve.

property `TrimmedCurve.end`: `ansys.geometry.core.math.point.Point3D`

End point of the curve.

property `TrimmedCurve.length`: `pint.Quantity`

Calculated length of the edge.

property `TrimmedCurve.interval`: `ansys.geometry.core.shapes.parameterization.Interval`

Interval of the curve that provides its boundary.

Method detail

`TrimmedCurve.evaluate_proportion(param: ansys.geometry.core.typing.Real) → ansys.geometry.core.shapes.curves.curve_evaluation.CurveEvaluation`

Evaluate the curve at a proportional value.

A parameter of 0 corresponds to the start of the curve, while a parameter of 1 corresponds to the end of the curve.

Parameters

param

[Real] Parameter in the proportional range [0,1].

Returns

CurveEvaluation

Resulting curve evaluation.

`TrimmedCurve.intersect_curve(other: TrimmedCurve) → list[ansys.geometry.core.math.point.Point3D]`

Get the intersect points of this trimmed curve with another one.

If the two trimmed curves do not intersect, an empty list is returned.

Parameters

other

[TrimmedCurve] Trimmed curve to intersect with.

Returns

list[Point3D]

All points of intersection between the curves.

`TrimmedCurve.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → TrimmedCurve`

Return a copy of the trimmed curve transformed by the given matrix.

Parameters

matrix

[Matrix44] Transformation matrix to apply to the curve.

Returns

TrimmedCurve

A new trimmed curve with the transformation applied.

`TrimmedCurve.translate(direction: ansys.geometry.core.math.vector.UnitVector3D, distance: ansys.geometry.core.typing.Real | pint.Quantity | ansys.geometry.core.misc.measurements.Distance) → None`

Translate the trimmed curve by a given vector and distance.

Parameters

direction

[UnitVector3D] Direction of translation.

distance

[Real | Quantity | Distance] Distance to translate.

`TrimmedCurve.rotate(origin: ansys.geometry.core.math.point.Point3D, axis: ansys.geometry.core.math.vector.UnitVector3D, angle: ansys.geometry.core.typing.Real | pint.Quantity | ansys.geometry.core.misc.measurements.Angle) → None`

Rotate the trimmed curve around a given axis centered at a given point.

Parameters

origin

[Point3D] Origin point of the rotation.

axis

[UnitVector3D] Axis of rotation.

angle

[Real | Quantity | Angle] Angle to rotate in radians.

TrimmedCurve.__repr__() → str

Represent the trimmed curve as a string.

ReversedTrimmedCurve

```
class ansys.geometry.core.shapes.curves.trimmed_curve.ReversedTrimmedCurve(geometry: an-  
sys.geometry.core.shapes.curves.curv  
start: an-  
sys.geometry.core.math.point.Point3D  
end: an-  
sys.geometry.core.math.point.Point3D  
interval: an-  
sys.geometry.core.shapes.parameteriz  
length:  
pint.Quantity,  
grpc_client: an-  
sys.geometry.core.connection.client.C  
= None)
```

Bases: TrimmedCurve

Represents a reversed trimmed curve.

When a curve is reversed, its start and end points are swapped, and parameters for evaluations are handled to provide expected results conforming to the direction of the curve. For example, evaluating a trimmed curve proportionally at 0 evaluates at the start point of the curve, but evaluating a reversed trimmed curve proportionally at 0 evaluates at what was previously the end point of the curve but is now the start point.

Parameters

geometry

[Curve] Underlying mathematical representation of the curve.

start

[Point3D] Original start point of the curve.

end

[Point3D] Original end point of the curve.

interval

[Interval] Parametric interval that bounds the curve.

length

[Quantity] Length of the curve.

grpc_client

[GrpcClient, default: `None`] gRPC client that is required for advanced functions such as `intersect_curve()`.

Overview

Methods

<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

<code>cone</code>	Provides for creating and managing a cone.
<code>cylinder</code>	Provides for creating and managing a cylinder.
<code>nurbs</code>	Provides for creating and managing a NURBS surface.
<code>plane</code>	Provides for creating and managing a plane.
<code>sphere</code>	Provides for creating and managing a sphere.
<code>surface</code>	Provides the Surface class.
<code>surface_evaluation</code>	Provides for evaluating a surface.
<code>torus</code>	Provides for creating and managing a torus.
<code>trimmed_surface</code>	Provides the TrimmedSurface class.

The cone.py module

Summary

Classes

<i>Cone</i>	Provides 3D cone representation.
<i>ConeEvaluation</i>	Evaluate the cone at given parameters.

Cone

```
class ansys.geometry.core.shapes-surfaces.cone.Cone(origin: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.point.Point3D, radius:
pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance |
ansys.geometry.core.typing.Real, half_angle:
pint.Quantity |
ansys.geometry.core.misc.measurements.Angle |
ansys.geometry.core.typing.Real, reference:
numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_X, axis: numpy.ndarray |
ansys.geometry.core.typing.RealSequence |
ansys.geometry.core.math.vector.UnitVector3D |
ansys.geometry.core.math.vector.Vector3D =
UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes-surfaces.surface.Surface*

Provides 3D cone representation.

Parameters

origin

[*ndarray* | *RealSequence* | *Point3D*] Origin of the cone.

radius

[*Quantity* | *Distance* | *Real*] Radius of the cone.

half_angle

[*Quantity* | *Angle* | *Real*] Half angle of the apex, determining the upward angle.

reference

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] X-axis direction.

axis

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] Z-axis direction.

Overview

Abstract methods

<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.

Methods

<i>transformed_copy</i>	Create a transformed copy of the cone from a transformation matrix.
<i>mirrored_copy</i>	Create a mirrored copy of the cone along the y-axis.
<i>evaluate</i>	Evaluate the cone at given parameters.
<i>project_point</i>	Project a point onto the cone and evaluate the cone.
<i>parameterization</i>	Parameterize the cone surface as a tuple (U and V respectively).

Properties

<i>origin</i>	Origin of the cone.
<i>radius</i>	Radius of the cone.
<i>half_angle</i>	Half angle of the apex.
<i>dir_x</i>	X-direction of the cone.
<i>dir_y</i>	Y-direction of the cone.
<i>dir_z</i>	Z-direction of the cone.
<i>height</i>	Height of the cone.
<i>surface_area</i>	Surface area of the cone.
<i>volume</i>	Volume of the cone.
<i>apex</i>	Apex point of the cone.
<i>apex_param</i>	Apex parameter of the cone.
<i>visualization_polydata</i>	Generate a visualization mesh for the cone.

Special methods

<i>__eq__</i>	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.

property `Cone.visualization_polydata`: `pyvista.PolyData`

Generate a visualization mesh for the cone.

Returns

pv.PolyData

Visualization mesh for the cone.

Method detail

`Cone.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Cone`

Create a transformed copy of the cone from a transformation matrix.

Parameters

matrix

[Matrix44] 4x4 transformation matrix to apply to the cone.

Returns

Cone

New cone that is the transformed copy of the original cone.

`Cone.mirrored_copy() → Cone`

Create a mirrored copy of the cone along the y-axis.

Returns

Cone

New cone that is a mirrored copy of the original cone.

`Cone.__eq__(other: Cone) → bool`

Equals operator for the Cone class.

`Cone.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → ConeEvaluation`

Evaluate the cone at given parameters.

Parameters

parameter

[ParamUV] Parameters (u,v) to evaluate the cone at.

Returns

ConeEvaluation

Resulting evaluation.

Cone.project_point(*point*: [ansys.geometry.core.math.point.Point3D](#)) → [ConeEvaluation](#)

Project a point onto the cone and evaluate the cone.

Parameters

point

[[Point3D](#)] Point to project onto the cone.

Returns

ConeEvaluation

Resulting evaluation.

Cone.parameterization() → [tuple](#)[[ansys.geometry.core.shapes.parameterization.Parameterization](#),
[ansys.geometry.core.shapes.parameterization.Parameterization](#)]

Parameterize the cone surface as a tuple (U and V respectively).

The U parameter specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the distance along the axis, with a zero parameter at the XY plane of the cone.

Returns

[tuple](#)[[Parameterization](#), [Parameterization](#)]

Information about how a cone's u and v parameters are parameterized, respectively.

abstractmethod **Cone.contains_param**(*param_uv*: [ansys.geometry.core.shapes.parameterization.ParamUV](#))
→ [bool](#)

Check a parameter is within the parametric range of the surface.

abstractmethod **Cone.contains_point**(*point*: [ansys.geometry.core.math.point.Point3D](#)) → [bool](#)

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

ConeEvaluation

class [ansys.geometry.core.shapes-surfaces-cone.ConeEvaluation](#)(*cone*: [Cone](#), *parameter*: [ansys.geometry.core.shapes.parameterization.ParamUV](#))

Bases: [ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation](#)

Evaluate the cone at given parameters.

Parameters

cone: [~ansys.geometry.core.shapes-surfaces-cone.Cone](#)

Cone to evaluate.

parameter: [ParamUV](#)

Parameters (u, v) to evaluate the cone at.

Overview

Properties

<code>cone</code>	Cone being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	Second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	Second derivative with respect to the V parameter.
<code>min_curvature</code>	Minimum curvature of the cone.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature of the cone.
<code>max_curvature_direction</code>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.cone import ConeEvaluation
```

Property detail

property `ConeEvaluation.cone:` `Cone`

Cone being evaluated.

property `ConeEvaluation.parameter:` `ansys.geometry.core.shapes.parameterization.ParamUV`

Parameter that the evaluation is based upon.

property `ConeEvaluation.position:` `ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

Point3D

Point that lies on the cone at this evaluation.

property `ConeEvaluation.normal:` `ansys.geometry.core.math.vector.UnitVector3D`

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the cone at this evaluation.

property `ConeEvaluation.u_derivative:` `ansys.geometry.core.math.vector.Vector3D`

First derivative with respect to the U parameter.

Returns

Vector3D

First derivative with respect to the U parameter.

property `ConeEvaluation.v_derivative:` `ansys.geometry.core.math.vector.Vector3D`

First derivative with respect to the V parameter.

Returns

Vector3D

First derivative with respect to the V parameter.

property ConeEvaluation.uu_derivative: [*ansys.geometry.core.math.vector.Vector3D*](#)

Second derivative with respect to the U parameter.

Returns**Vector3D**

Second derivative with respect to the U parameter.

property ConeEvaluation.uv_derivative: [*ansys.geometry.core.math.vector.Vector3D*](#)

Second derivative with respect to the U and V parameters.

Returns**Vector3D**

Second derivative with respect to U and V parameters.

property ConeEvaluation.vv_derivative: [*ansys.geometry.core.math.vector.Vector3D*](#)

Second derivative with respect to the V parameter.

Returns**Vector3D**

Second derivative with respect to the V parameter.

property ConeEvaluation.min_curvature: [*ansys.geometry.core.typing.Real*](#)

Minimum curvature of the cone.

Returns**Real**

Minimum curvature of the cone.

property ConeEvaluation.min_curvature_direction:

[*ansys.geometry.core.math.vector.UnitVector3D*](#)

Minimum curvature direction.

Returns**UnitVector3D**

Minimum curvature direction.

property ConeEvaluation.max_curvature: [*ansys.geometry.core.typing.Real*](#)

Maximum curvature of the cone.

Returns**Real**

Maximum curvature of the cone.

property ConeEvaluation.max_curvature_direction:

[*ansys.geometry.core.math.vector.UnitVector3D*](#)

Maximum curvature direction.

Returns**UnitVector3D**

Maximum curvature direction.

Description

Provides for creating and managing a cone.

The `cylinder.py` module

Summary

Classes

<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: numpy.ndarray | an-
sys.geometry.core.typing.RealSequence
| an-
sys.geometry.core.math.point.Point3D,
radius: pint.Quantity | an-
sys.geometry.core.misc.measurements.Distance
| ansys.geometry.core.typing.Real,
reference: numpy.ndarray | an-
sys.geometry.core.typing.RealSequence
| an-
sys.geometry.core.math.vector.UnitVector3D
| an-
sys.geometry.core.math.vector.Vector3D
= UNITVECTOR3D_X, axis:
numpy.ndarray | an-
sys.geometry.core.typing.RealSequence
| an-
sys.geometry.core.math.vector.UnitVector3D
| an-
sys.geometry.core.math.vector.Vector3D
= UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes-surfaces.surface.Surface*

Provides 3D cylinder representation.

Parameters

origin

[*ndarray* | *RealSequence* | *Point3D*] Origin of the cylinder.

radius

[*Quantity* | *Distance* | *Real*] Radius of the cylinder.

reference

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] X-axis direction.

axis

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] Z-axis direction.

Overview

Abstract methods

<code>contains_param</code>	Check a parameter is within the parametric range of the surface.
<code>contains_point</code>	Check a point is contained by the surface.

Methods

<code>surface_area</code>	Get the surface area of the cylinder.
<code>volume</code>	Get the volume of the cylinder.
<code>transformed_copy</code>	Create a transformed copy of the cylinder from a transformation matrix.
<code>mirrored_copy</code>	Create a mirrored copy of the cylinder along the y-axis.
<code>evaluate</code>	Evaluate the cylinder at the given parameters.
<code>project_point</code>	Project a point onto the cylinder and evaluate the cylinder.
<code>parameterization</code>	Parameterize the cylinder surface as a tuple (U and V respectively).

Properties

<code>origin</code>	Origin of the cylinder.
<code>radius</code>	Radius of the cylinder.
<code>dir_x</code>	X-direction of the cylinder.
<code>dir_y</code>	Y-direction of the cylinder.
<code>dir_z</code>	Z-direction of the cylinder.
<code>visualization_polydata</code>	Get the visualization polydata for 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.

property `Cylinder.visualization_polydata`: `pyvista.PolyData`

Get the visualization polydata for the cylinder.

Returns

`pyvista.PolyData`

Polydata for visualizing the cylinder.

Notes

The cylinder is infinite by nature, so this visualization is just a finite representation of the cylinder for visualization purposes. The height of the visualized cylinder is arbitrarily set to 2 times the radius.

Method detail

`Cylinder.surface_area`(*height*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`) → `pint.Quantity`

Get the surface area of the cylinder.

Parameters

height

[`Quantity` | `Distance` | `Real`] Height to bound the cylinder at.

Returns

`Quantity`

Surface area of the temporarily bounded cylinder.

Notes

By nature, a cylinder is infinite. If you want to get the surface area, you must bound it by a height. Normally a cylinder surface is not closed (does not have “caps” on the ends). This method assumes that the cylinder is closed for the purpose of getting the surface area.

`Cylinder.volume`(*height*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`) → `pint.Quantity`

Get the volume of the cylinder.

Parameters

height

[`Quantity` | `Distance` | `Real`] Height to bound the cylinder at.

Returns

`Quantity`

Volume of the temporarily bounded cylinder.

Notes

By nature, a cylinder is infinite. If you want to get the surface area, you must bound it by a height. Normally a cylinder surface is not closed (does not have “caps” on the ends). This method assumes that the cylinder is closed for the purpose of getting the surface area.

`Cylinder.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Cylinder`

Create a transformed copy of the cylinder from a transformation matrix.

Parameters

matrix

[Matrix44] 4X4 transformation matrix to apply to the cylinder.

Returns

Cylinder

New cylinder that is the transformed copy of the original cylinder.

`Cylinder.mirrored_copy() → Cylinder`

Create a mirrored copy of the cylinder along the y-axis.

Returns

Cylinder

New cylinder that is a mirrored copy of the original cylinder.

`Cylinder.__eq__(other: Cylinder) → bool`

Equals operator for the *Cylinder* class.

`Cylinder.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → CylinderEvaluation`

Evaluate the cylinder at the given parameters.

Parameters

parameter

[ParamUV] Parameters (u,v) to evaluate the cylinder at.

Returns

CylinderEvaluation

Resulting evaluation.

`Cylinder.project_point(point: ansys.geometry.core.math.point.Point3D) → CylinderEvaluation`

Project a point onto the cylinder and evaluate the cylinder.

Parameters

point

[Point3D] Point to project onto the cylinder.

Returns

CylinderEvaluation

Resulting evaluation.

`Cylinder.parameterization() → tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Parameterize the cylinder surface as a tuple (U and V respectively).

The U parameter specifies the clockwise angle around the axis (right-hand corkscrew law), with a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the distance along the axis, with a zero parameter at the XY plane of the cylinder.

Returns

tuple[Parameterization, Parameterization]

Information about how a cylinder's u and v parameters are parameterized, respectively.

abstractmethod `Cylinder.contains_param`(*param_uv*: `ansys.geometry.core.shapes.parameterization.ParamUV`) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod `Cylinder.contains_point`(*point*: `ansys.geometry.core.math.point.Point3D`) → bool

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

CylinderEvaluation

class `ansys.geometry.core.shapes-surfaces.cylinder.CylinderEvaluation`(*cylinder*: `Cylinder`,
parameter: `ansys.geometry.core.shapes.parameterization.ParamUV`)

Bases: `ansys.geometry.core.shapes-surfaces.surface_evaluation.SurfaceEvaluation`

Provides evaluation of a cylinder at given parameters.

Parameters

cylinder: `~ansys.geometry.core.shapes-surfaces.cylinder.Cylinder`

Cylinder to evaluate.

parameter: `ParamUV`

Parameters (u, v) to evaluate the cylinder at.

Overview

Properties

<i>cylinder</i>	Cylinder being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	Second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	Second derivative with respect to the V parameter.
<i>min_curvature</i>	Minimum curvature of the cylinder.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature of the cylinder.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.cylinder import CylinderEvaluation
```

Property detail

property `CylinderEvaluation.cylinder`: `Cylinder`

Cylinder being evaluated.

property `CylinderEvaluation.parameter:`

`ansys.geometry.core.shapes.parameterization.ParamUV`

Parameter that the evaluation is based upon.

property `CylinderEvaluation.position:` `ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

Point3D

Point that lies on the cylinder at this evaluation.

property `CylinderEvaluation.normal:` `ansys.geometry.core.math.vector.UnitVector3D`

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the cylinder at this evaluation.

property `CylinderEvaluation.u_derivative:` `ansys.geometry.core.math.vector.Vector3D`

First derivative with respect to the U parameter.

Returns

Vector3D

First derivative with respect to the U parameter.

property `CylinderEvaluation.v_derivative:` `ansys.geometry.core.math.vector.Vector3D`

First derivative with respect to the V parameter.

Returns

Vector3D

First derivative with respect to the V parameter.

property `CylinderEvaluation.uu_derivative:` `ansys.geometry.core.math.vector.Vector3D`

Second derivative with respect to the U parameter.

Returns

Vector3D

Second derivative with respect to the U parameter.

property `CylinderEvaluation.uv_derivative:` `ansys.geometry.core.math.vector.Vector3D`

Second derivative with respect to the U and V parameters.

Returns

Vector3D

Second derivative with respect to the U and v parameters.

property `CylinderEvaluation.vv_derivative:` `ansys.geometry.core.math.vector.Vector3D`

Second derivative with respect to the V parameter.

Returns

Vector3D

Second derivative with respect to the V parameter.

property `CylinderEvaluation.min_curvature:` [*ansys.geometry.core.typing.Real*](#)

Minimum curvature of the cylinder.

Returns

Real

Minimum curvature of the cylinder.

property `CylinderEvaluation.min_curvature_direction:`
[*ansys.geometry.core.math.vector.UnitVector3D*](#)

Minimum curvature direction.

Returns

UnitVector3D

Minimum curvature direction.

property `CylinderEvaluation.max_curvature:` [*ansys.geometry.core.typing.Real*](#)

Maximum curvature of the cylinder.

Returns

Real

Maximum curvature of the cylinder.

property `CylinderEvaluation.max_curvature_direction:`
[*ansys.geometry.core.math.vector.UnitVector3D*](#)

Maximum curvature direction.

Returns

UnitVector3D

Maximum curvature direction.

Description

Provides for creating and managing a cylinder.

The `nurbs.py` module

Summary

Classes

<i>NURBSSurface</i>	Represents a NURBS surface.
<i>NURBSSurfaceEvaluation</i>	Provides evaluation of a NURBS surface at a given parameter.

NURBSSurface

```
class ansys.geometry.core.shapes-surfaces.nurbs.NURBSSurface(origin:  
    ansys.geometry.core.math.Point3D =  
    ZERO_POINT3D, reference: an-  
    sys.geometry.core.math.vector.UnitVector3D  
    = UNITVECTOR3D_X, axis: an-  
    sys.geometry.core.math.vector.UnitVector3D  
    = UNITVECTOR3D_Z,  
    geomdl_object:  
    geomdl.NURBS.Surface = None)
```

Bases: `ansys.geometry.core.shapes-surfaces.surface.Surface`

Represents a NURBS surface.

Notes

This class is a wrapper around the NURBS surface class from the `geomdl` library. By leveraging the `geomdl` library, this class provides a high-level interface to create and manipulate NURBS surfaces. The `geomdl` library is a powerful library for working with NURBS curves and surfaces. For more information, see <https://pypi.org/project/geomdl/>.

Overview

Abstract methods

<code>transformed_copy</code>	Create a transformed copy of the surface.
<code>contains_param</code>	Check a parameter is within the parametric range of the surface.
<code>contains_point</code>	Check a point is contained by the surface.
<code>project_point</code>	Project a point to the surface.

Constructors

<code>from_control_points</code>	Create a NURBS surface from control points and knot vectors.
<code>fit_surface_from_points</code>	Fit a NURBS surface to a set of points.

Methods

<code>parameterization</code>	Get the parametrization of the NURBS surface.
<code>evaluate</code>	Evaluate the surface at the given parameter.

Properties

<code>geomdl_nurbs_surface</code>	Get the underlying <code>geomdl</code> NURBS surface object.
<code>control_points</code>	Get the control points of the NURBS surface.
<code>degree_u</code>	Get the degree of the surface in the U direction.
<code>degree_v</code>	Get the degree of the surface in the V direction.
<code>knotvector_u</code>	Get the knot vector for the u-direction of the surface.
<code>knotvector_v</code>	Get the knot vector for the v-direction of the surface.
<code>weights</code>	Get the weights of the control points.
<code>origin</code>	Get the origin of the surface.
<code>dir_x</code>	Get the reference direction of the surface.
<code>dir_z</code>	Get the axis direction of the surface.
<code>visualization_polydata</code>	Get the visualization polydata for the NURBS surface.

Special methods

<code>__eq__</code>	Determine if two surfaces are equal.
---------------------	--------------------------------------

Import detail

```
from ansys.geometry.core.shapes-surfaces.nurbs import NURBSSurface
```

Property detail

property NURBSSurface.geomdl_nurbs_surface: geomdl.NURBS.Surface

Get the underlying *geomdl* NURBS surface object.

Notes

This property gives access to the full functionality of the NURBS surface coming from the *geomdl* library. Use with caution.

property NURBSSurface.control_points: list[ansys.geometry.core.math.Point3D]

Get the control points of the NURBS surface.

property NURBSSurface.degree_u: int

Get the degree of the surface in the U direction.

property NURBSSurface.degree_v: int

Get the degree of the surface in the V direction.

property NURBSSurface.knotvector_u: list[ansys.geometry.core.typing.Real]

Get the knot vector for the u-direction of the surface.

property NURBSSurface.knotvector_v: list[ansys.geometry.core.typing.Real]

Get the knot vector for the v-direction of the surface.

property NURBSSurface.weights: list[ansys.geometry.core.typing.Real]

Get the weights of the control points.

property NURBSSurface.origin: ansys.geometry.core.math.Point3D

Get the origin of the surface.

property NURBSSurface.dir_x: ansys.geometry.core.math.vector.UnitVector3D

Get the reference direction of the surface.

property NURBSSurface.dir_z: ansys.geometry.core.math.vector.UnitVector3D

Get the axis direction of the surface.

property NURBSSurface.visualization_polydata: pyvista.PolyData

Get the visualization polydata for the NURBS surface.

This method generates a triangulated mesh representation of the NURBS surface by sampling it at a grid of UV parameters and creating a structured mesh. The resolution is automatically determined based on the surface complexity, with a minimum of 20 points per direction and additional sampling for higher-degree surfaces.

Returns

pv.PolyData

Triangulated mesh representation of the NURBS surface.

Notes

The surface is sampled in the parametric domain $[u_{\min}, u_{\max}] \times [v_{\min}, v_{\max}]$ and evaluated at a grid of points. The resulting mesh respects the surface's local coordinate system defined by origin, reference (*dir_x*), and axis (*dir_z*).

Method detail

```
classmethod NURBSSurface.from_control_points(degree_u: int, degree_v: int, knots_u:
                                             list[ansys.geometry.core.typing.Real], knots_v:
                                             list[ansys.geometry.core.typing.Real], control_points:
                                             list[ansys.geometry.core.math.Point3D], weights:
                                             list[ansys.geometry.core.typing.Real] = None, origin:
                                             ansys.geometry.core.math.Point3D = ZERO_POINT3D,
                                             reference: ansys.geometry.core.math.vector.UnitVector3D =
                                             UNITVECTOR3D_X, axis:
                                             ansys.geometry.core.math.vector.UnitVector3D =
                                             UNITVECTOR3D_Z) → NURBSSurface
```

Create a NURBS surface from control points and knot vectors.

Parameters

degree_u
[int] Degree of the surface in the U direction.

degree_v
[int] Degree of the surface in the V direction.

knots_u
[list[Real]] Knot vector for the U direction.

knots_v
[list[Real]] Knot vector for the V direction.

control_points
[list[Point3D]] Control points for the surface.

weights
[list[Real], optional] Weights for the control points. If not provided, all weights are set to 1.

delta
[float, optional] Evaluation delta for the surface. Default is 0.01.

origin
[Point3D, optional] Origin of the surface. Default is (0, 0, 0).

reference
[UnitVector3D, optional] Reference direction of the surface. Default is (1, 0, 0).

axis
[UnitVector3D, optional] Axis direction of the surface. Default is (0, 0, 1).

Returns

NURBSSurface
Created NURBS surface.

```
classmethod NURBSSurface.fit_surface_from_points(points: list[ansys.geometry.core.math.Point3D],
                                                  size_u: int, size_v: int, degree_u: int, degree_v: int,
                                                  origin: ansys.geometry.core.math.Point3D =
                                                  ZERO_POINT3D, reference:
                                                  ansys.geometry.core.math.vector.UnitVector3D =
                                                  UNITVECTOR3D_X, axis:
                                                  ansys.geometry.core.math.vector.UnitVector3D =
                                                  UNITVECTOR3D_Z) → NURBSSurface
```

Fit a NURBS surface to a set of points.

Parameters**points**

[[list](#)[[Point3D](#)]] Points to fit the surface to.

size_u

[[int](#)] Number of control points in the U direction.

size_v

[[int](#)] Number of control points in the V direction.

degree_u

[[int](#)] Degree of the surface in the U direction.

degree_v

[[int](#)] Degree of the surface in the V direction.

origin

[[Point3D](#), optional] Origin of the surface. Default is (0, 0, 0).

reference

[[UnitVector3D](#), optional] Reference direction of the surface. Default is (1, 0, 0).

axis

[[UnitVector3D](#), optional] Axis direction of the surface. Default is (0, 0, 1).

Returns**NURBSSurface**

Fitted NURBS surface.

`NURBSSurface.__eq__(other: NURBSSurface) → bool`

Determine if two surfaces are equal.

`NURBSSurface.parameterization() → tuple[ansys.geometry.core.shapes.parameterization.Parameterization, ansys.geometry.core.shapes.parameterization.Parameterization]`

Get the parametrization of the NURBS surface.

The parameter is defined in the interval [0, 1] by default. Information is provided about the parameter type and form.

Returns

`tuple[Parameterization, Parameterization]`

Parameterization in the U and V directions respectively.

`abstractmethod NURBSSurface.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → NURBSSurface`

Create a transformed copy of the surface.

`NURBSSurface.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → ansys.geometry.core.shapes.surfaces.surface_evaluation.SurfaceEvaluation`

Evaluate the surface at the given parameter.

Parameters**parameter**

[[ParamUV](#)] Parameter to evaluate the surface at.

Returns**SurfaceEvaluation**

Evaluation of the surface at the given parameter.

abstractmethod `NURBSSurface.contains_param`(*param*: `ansys.geometry.core.shapes.parameterization.ParamUV`) → `bool`

Check a parameter is within the parametric range of the surface.

abstractmethod `NURBSSurface.contains_point`(*point*: `ansys.geometry.core.math.Point3D`) → `bool`

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

abstractmethod `NURBSSurface.project_point`(*point*: `ansys.geometry.core.math.Point3D`) → `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`

Project a point to the surface.

This method returns the evaluation at the closest point.

NURBSSurfaceEvaluation

class `ansys.geometry.core.shapes-surfaces-nurbs.NURBSSurfaceEvaluation`(*nurbs_surface*: `NURBSSurface`, *parameter*: `ansys.geometry.core.shapes-parameterization`)

Bases: `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`

Provides evaluation of a NURBS surface at a given parameter.

Parameters

nurbs_surface: `~ansys.geometry.core.shapes-surfaces-nurbs.NURBSSurface`
NURBS surface to evaluate.

parameter: `Real`
Parameter to evaluate the NURBS surface at.

Overview

Properties

<code>surface</code>	Surface being evaluated.
<code>parameter</code>	Parameter the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	The second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	The second derivative with respect to v.
<code>min_curvature</code>	Minimum curvature.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature.
<code>max_curvature_direction</code>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.nurbs import NURBSSurfaceEvaluation
```

Property detail

property NURBSSurfaceEvaluation.surface: *NURBSSurface*

Surface being evaluated.

property NURBSSurfaceEvaluation.parameter:
ansys.geometry.core.shapes.parameterization.ParamUV

Parameter the evaluation is based upon.

property NURBSSurfaceEvaluation.position: *ansys.geometry.core.math.Point3D*

Position of the evaluation.

Returns

Point3D

Point on the surface at this evaluation.

property NURBSSurfaceEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

Returns

UnitVector3D

Normal to the surface at this evaluation.

property NURBSSurfaceEvaluation.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 at this evaluation.

property NURBSSurfaceEvaluation.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 at this evaluation.

property NURBSSurfaceEvaluation.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 at this evaluation.

property NURBSSurfaceEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

The second derivative with respect to the U and V parameters.

Returns

Vector3D

Second derivative with respect to the U and V parameters at this evaluation.

property NURBSSurfaceEvaluation.vv_derivative: *ansys.geometry.core.math.vector.Vector3D*

The second derivative with respect to v.

Returns

Vector3D

Second derivative with respect to the V parameter at this evaluation.

property NURBSSurfaceEvaluation.min_curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Minimum curvature.

property NURBSSurfaceEvaluation.min_curvature_direction:

ansys.geometry.core.math.vector.UnitVector3D

Abstractmethod

Minimum curvature direction.

property NURBSSurfaceEvaluation.max_curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Maximum curvature.

property NURBSSurfaceEvaluation.max_curvature_direction:

ansys.geometry.core.math.vector.UnitVector3D

Abstractmethod

Maximum curvature direction.

Description

Provides for creating and managing a NURBS surface.

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: numpy.ndarray | an-
sys.geometry.core.typing.RealSequence
| an-
sys.geometry.core.math.point.Point3D,
reference: numpy.ndarray | an-
sys.geometry.core.typing.RealSequence
| an-
sys.geometry.core.math.vector.UnitVector3D
| an-
sys.geometry.core.math.vector.Vector3D
= UNITVECTOR3D_X, axis:
numpy.ndarray | an-
sys.geometry.core.typing.RealSequence
| an-
sys.geometry.core.math.vector.UnitVector3D
| an-
sys.geometry.core.math.vector.Vector3D
= UNITVECTOR3D_Z)

```

Bases: *ansys.geometry.core.shapes.surfaces.surface.Surface*

Provides 3D plane surface representation.

Parameters

origin

[*ndarray* | *RealSequence* | *Point3D*] Centered origin of the plane.

reference

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] X-axis direction.

axis

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] Z-axis direction.

Overview

Abstract methods

<i>contains_param</i>	Check a ParamUV is within the parametric range of the surface.
<i>contains_point</i>	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.
<i>evaluate</i>	Evaluate the plane at a given u and v parameter.

Properties

<code>origin</code>	Origin of the plane.
<code>dir_x</code>	X-direction of the plane.
<code>dir_y</code>	Y-direction of the plane.
<code>dir_z</code>	Z-direction of the plane.
<code>visualization_polydata</code>	Get the visualization polydata for the plane.

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 plane.

property `PlaneSurface.dir_x`: `ansys.geometry.core.math.vector.UnitVector3D`

X-direction of the plane.

property `PlaneSurface.dir_y`: `ansys.geometry.core.math.vector.UnitVector3D`

Y-direction of the plane.

property `PlaneSurface.dir_z`: `ansys.geometry.core.math.vector.UnitVector3D`

Z-direction of the plane.

property `PlaneSurface.visualization_polydata`: `pyvista.PolyData`

Get the visualization polydata for the plane.

Returns

pv.PolyData

Polydata representation of the plane for visualization.

Method detail

`PlaneSurface.__eq__(other: PlaneSurface) → bool`

Check whether two planes are equal.

abstractmethod `PlaneSurface.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`

Check a `ParamUV` is within the parametric range of the surface.

abstractmethod `PlaneSurface.contains_point(point: ansys.geometry.core.math.point.Point3D) → bool`

Check whether a 3D point is in the domain of the plane.

`PlaneSurface.parameterization()` → tuple[*ansys.geometry.core.shapes.parameterization.Parameterization*,
ansys.geometry.core.shapes.parameterization.Parameterization]

Parametrize the plane.

`PlaneSurface.project_point(point: ansys.geometry.core.math.point.Point3D)` →
ansys.geometry.core.shapes-surfaces.surface_evaluation.SurfaceEvaluation

Evaluate the plane at a given 3D point.

`PlaneSurface.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44)` →
ansys.geometry.core.shapes-surfaces.surface.Surface

Get a transformed version of the plane given the transform.

`PlaneSurface.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV)` →
PlaneEvaluation

Evaluate the plane at a given u and v parameter.

PlaneEvaluation

class `ansys.geometry.core.shapes-surfaces.plane.PlaneEvaluation(plane: PlaneSurface, parameter: an-
sys.geometry.core.shapes.parameterization.ParamUV`

Bases: *ansys.geometry.core.shapes-surfaces.surface_evaluation.SurfaceEvaluation*

Provides evaluation of a plane at given parameters.

Parameters

plane: ~`ansys.geometry.core.shapes-surfaces.plane.PlaneSurface`
Plane to evaluate.

parameter: `ParamUV`
Parameters (u, v) to evaluate the plane at.

Overview

Properties

<i>plane</i>	Plane being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Point on the surface, based on the evaluation.
<i>normal</i>	Normal to the surface.
<i>u_derivative</i>	First derivative with respect to u.
<i>v_derivative</i>	First derivative with respect to v.
<i>uu_derivative</i>	Second derivative with respect to u.
<i>uv_derivative</i>	Second derivative with respect to u and v.
<i>vv_derivative</i>	Second derivative with respect to v.
<i>min_curvature</i>	Minimum curvature.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.plane import PlaneEvaluation
```

Property detail

property PlaneEvaluation.plane: *PlaneSurface*

Plane being evaluated.

property PlaneEvaluation.parameter: *ansys.geometry.core.shapes.parameterization.ParamUV*

Parameter that the evaluation is based upon.

property PlaneEvaluation.position: *ansys.geometry.core.math.point.Point3D*

Point on the surface, based on the evaluation.

property PlaneEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

Normal to the surface.

property PlaneEvaluation.u_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to u.

property PlaneEvaluation.v_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to v.

property PlaneEvaluation.uu_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to u.

property PlaneEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to u and v.

property PlaneEvaluation.vv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to v.

property PlaneEvaluation.min_curvature: *ansys.geometry.core.typing.Real*

Minimum curvature.

property PlaneEvaluation.min_curvature_direction:
ansys.geometry.core.math.vector.UnitVector3D

Minimum curvature direction.

property PlaneEvaluation.max_curvature: *ansys.geometry.core.typing.Real*

Maximum curvature.

property PlaneEvaluation.max_curvature_direction:
ansys.geometry.core.math.vector.UnitVector3D

Maximum curvature direction.

Description

Provides for creating and managing a plane.

The sphere.py module

Summary

Classes

<i>Sphere</i>	Provides 3D sphere representation.
<i>SphereEvaluation</i>	Evaluate a sphere at given parameters.

Sphere

```
class ansys.geometry.core.shapes.sphere.Sphere(origin: numpy.ndarray |  
ansys.geometry.core.typing.RealSequence |  
ansys.geometry.core.math.point.Point3D,  
radius: pint.Quantity | an-  
sys.geometry.core.misc.measurements.Distance  
| ansys.geometry.core.typing.Real,  
reference: numpy.ndarray |  
ansys.geometry.core.typing.RealSequence |  
an-  
sys.geometry.core.math.vector.UnitVector3D  
|  
ansys.geometry.core.math.vector.Vector3D  
= UNITVECTOR3D_X, axis:  
numpy.ndarray |  
ansys.geometry.core.typing.RealSequence |  
an-  
sys.geometry.core.math.vector.UnitVector3D  
|  
ansys.geometry.core.math.vector.Vector3D  
= UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes.surface.Surface*

Provides 3D sphere representation.

Parameters

origin

[*ndarray* | *RealSequence* | *Point3D*] Origin of the sphere.

radius

[*Quantity* | *Distance* | *Real*] Radius of the sphere.

reference

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] X-axis direction.

axis

[*ndarray* | *RealSequence* | *UnitVector3D* | *Vector3D*] Z-axis direction.

Overview

Abstract methods

<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.

Methods

<code>transformed_copy</code>	Create a transformed copy of the sphere from 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, V).

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.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

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.

property `Sphere.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

`Sphere.__eq__(other: Sphere) → bool`

Equals operator for the `Sphere` class.

`Sphere.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Sphere`

Create a transformed copy of the sphere from a transformation matrix.

Parameters

matrix

[Matrix44] 4X4 transformation matrix to apply to the sphere.

Returns

Sphere

New sphere that is the transformed copy of the original sphere.

`Sphere.mirrored_copy() → Sphere`

Create a mirrored copy of the sphere along the y-axis.

Returns

Sphere

New sphere that is a mirrored copy of the original sphere.

`Sphere.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV) → SphereEvaluation`

Evaluate the sphere at the given parameters.

Parameters

parameter

[ParamUV] Parameters (u,v) to evaluate the sphere at.

Returns

SphereEvaluation

Resulting evaluation.

`Sphere.project_point(point: ansys.geometry.core.math.point.Point3D) → SphereEvaluation`

Project a point onto the sphere and evaluate the sphere.

Parameters

point

[Point3D] Point to project onto the sphere.

Returns

SphereEvaluation

Resulting evaluation.

`Sphere.parameterization()` → tuple[*ansys.geometry.core.shapes.parameterization.Parameterization*,
ansys.geometry.core.shapes.parameterization.Parameterization]

Parameterization of the sphere surface as a tuple (U, V).

The U parameter specifies the longitude angle, increasing clockwise (east) about `dir_z` (right-hand corkscrew law). It has a zero parameter at `dir_x` and a period of 2π .

The V parameter specifies the latitude, increasing north, with a zero parameter at the equator and a range of $[-\pi/2, \pi/2]$.

Returns

tuple[*Parameterization*, *Parameterization*]

Information about how a sphere's u and v parameters are parameterized, respectively.

abstractmethod `Sphere.contains_param`(*param_uv*: *ansys.geometry.core.shapes.parameterization.ParamUV*) → bool

Check a parameter is within the parametric range of the surface.

abstractmethod `Sphere.contains_point`(*point*: *ansys.geometry.core.math.point.Point3D*) → bool

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

SphereEvaluation

class `ansys.geometry.core.shapes.surfaces.sphere.SphereEvaluation`(*sphere*: *Sphere*, *parameter*:
ansys.geometry.core.shapes.parameterization.ParamUV)

Bases: *ansys.geometry.core.shapes.surfaces.surface_evaluation.SurfaceEvaluation*

Evaluate a sphere at given parameters.

Parameters

sphere: *~ansys.geometry.core.shapes.surfaces.sphere.Sphere*
Sphere to evaluate.

parameter: *ParamUV*
Parameters (u, v) to evaluate the sphere at.

Overview

Properties

<i>sphere</i>	Sphere being evaluated.
<i>parameter</i>	Parameter that the evaluation is based upon.
<i>position</i>	Position of the evaluation.
<i>normal</i>	The normal to the surface.
<i>u_derivative</i>	First derivative with respect to the U parameter.
<i>v_derivative</i>	First derivative with respect to the V parameter.
<i>uu_derivative</i>	Second derivative with respect to the U parameter.
<i>uv_derivative</i>	Second derivative with respect to the U and V parameters.
<i>vv_derivative</i>	Second derivative with respect to the V parameter.
<i>min_curvature</i>	Minimum curvature of the sphere.
<i>min_curvature_direction</i>	Minimum curvature direction.
<i>max_curvature</i>	Maximum curvature of the sphere.
<i>max_curvature_direction</i>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.sphere import SphereEvaluation
```

Property detail

property SphereEvaluation.sphere: *Sphere*

Sphere being evaluated.

property SphereEvaluation.parameter: *ansys.geometry.core.shapes.parameterization.ParamUV*

Parameter that the evaluation is based upon.

property SphereEvaluation.position: *ansys.geometry.core.math.point.Point3D*

Position of the evaluation.

Returns

Point3D

Point that lies on the sphere at this evaluation.

property SphereEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

The normal to the surface.

Returns

UnitVector3D

Normal unit vector to the sphere at this evaluation.

property SphereEvaluation.u_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the U parameter.

Returns

Vector3D

First derivative with respect to the U parameter.

property SphereEvaluation.v_derivative: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the V parameter.

Returns

Vector3D

First derivative with respect to the V parameter.

property SphereEvaluation.uu_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U parameter.

Returns

Vector3D

Second derivative with respect to the U parameter.

property SphereEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U and V parameters.

Returns

Vector3D

The second derivative with respect to the U and V parameters.

property SphereEvaluation.vv_derivative: [ansys.geometry.core.math.vector.Vector3D](#)

Second derivative with respect to the V parameter.

Returns

Vector3D

The second derivative with respect to the V parameter.

property SphereEvaluation.min_curvature: [ansys.geometry.core.typing.Real](#)

Minimum curvature of the sphere.

Returns

Real

Minimum curvature of the sphere.

property SphereEvaluation.min_curvature_direction:

[ansys.geometry.core.math.vector.UnitVector3D](#)

Minimum curvature direction.

Returns

UnitVector3D

Minimum curvature direction.

property SphereEvaluation.max_curvature: [ansys.geometry.core.typing.Real](#)

Maximum curvature of the sphere.

Returns

Real

Maximum curvature of the sphere.

property SphereEvaluation.max_curvature_direction:

[ansys.geometry.core.math.vector.UnitVector3D](#)

Maximum curvature direction.

Returns

UnitVector3D

Maximum curvature direction.

Description

Provides for creating and managing a sphere.

The surface.py module

Summary

Classes

Surface	Provides the abstract base class for a 3D surface.
-------------------------	--

Surface

class ansys.geometry.core.shapes.surfaces.surface.Surface

Bases: [abc.ABC](#)

Provides the abstract base class for a 3D surface.

Overview

Abstract methods

<code>parameterization</code>	Parameterize the surface as a tuple (U and V respectively).
<code>contains_param</code>	Check a parameter is within the parametric range of the surface.
<code>contains_point</code>	Check a point is contained by the surface.
<code>transformed_copy</code>	Create a transformed copy of the surface.
<code>__eq__</code>	Determine if two surfaces are equal.
<code>evaluate</code>	Evaluate the surface at the given parameter.
<code>project_point</code>	Project a point to the surface.
<code>visualization_polydata</code>	Get the visualization polydata for the surface.

Methods

<code>trim</code>	Trim this surface by bounding it with a BoxUV.
-------------------	--

Import detail

```
from ansys.geometry.core.shapes-surfaces.surface import Surface
```

Method detail

abstractmethod `Surface.parameterization()` → tuple[`ansys.geometry.core.shapes.parameterization.Parameterization`, `ansys.geometry.core.shapes.parameterization.Parameterization`]

Parameterize the surface as a tuple (U and V respectively).

abstractmethod `Surface.contains_param(param_uv: ansys.geometry.core.shapes.parameterization.ParamUV)` → bool

Check a parameter is within the parametric range of the surface.

abstractmethod `Surface.contains_point(point: ansys.geometry.core.math.point.Point3D)` → bool

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

abstractmethod `Surface.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44)` → `Surface`

Create a transformed copy of the surface.

abstractmethod `Surface.__eq__(other: Surface)` → bool

Determine if two surfaces are equal.

abstractmethod `Surface.evaluate(parameter: ansys.geometry.core.shapes.parameterization.ParamUV)` → `ansys.geometry.core.shapes-surfaces.surface_evaluation.SurfaceEvaluation`

Evaluate the surface at the given parameter.

abstractmethod `Surface.project_point(point: ansys.geometry.core.math.point.Point3D)` → `ansys.geometry.core.shapes-surfaces.surface_evaluation.SurfaceEvaluation`

Project a point to the surface.

This method returns the evaluation at the closest point.

abstractmethod `Surface.visualization_polydata()` → `pyvista.PolyData`

Get the visualization polydata for the surface.

`Surface.trim(box_uv: ansys.geometry.core.shapes.box_uv.BoxUV)` →
`ansys.geometry.core.shapes-surfaces-trimmed_surface.TrimmedSurface`

Trim this surface by bounding it with a BoxUV.

Returns

TrimmedSurface

The resulting bounded surface.

Description

Provides the Surface class.

The surface_evaluation.py module

Summary

Classes

<code>SurfaceEvaluation</code>	Provides for evaluating a surface.
--------------------------------	------------------------------------

SurfaceEvaluation

class `ansys.geometry.core.shapes-surfaces-surface_evaluation.SurfaceEvaluation`(*parameter:*
`ansys.geometry.core.shapes-param`

Provides for evaluating a surface.

Overview

Properties

<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Point on the surface, based on 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>	The second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	The second derivative with respect to v.
<code>min_curvature</code>	Minimum curvature.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature.
<code>max_curvature_direction</code>	Maximum curvature direction.

Import detail

```
from ansys.geometry.core.shapes-surfaces.surface_evaluation import SurfaceEvaluation
```

Property detail

property SurfaceEvaluation.parameter: *ansys.geometry.core.shapes.parameterization.ParamUV*

Abstractmethod

Parameter that the evaluation is based upon.

property SurfaceEvaluation.position: *ansys.geometry.core.math.point.Point3D*

Abstractmethod

Point on the surface, based on the evaluation.

property SurfaceEvaluation.normal: *ansys.geometry.core.math.vector.UnitVector3D*

Abstractmethod

Normal to the surface.

property SurfaceEvaluation.u_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

First derivative with respect to the U parameter.

property SurfaceEvaluation.v_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

First derivative with respect to the V parameter.

property SurfaceEvaluation.uu_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

Second derivative with respect to the U parameter.

property SurfaceEvaluation.uv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

The second derivative with respect to the U and V parameters.

property SurfaceEvaluation.vv_derivative: *ansys.geometry.core.math.vector.Vector3D*

Abstractmethod

The second derivative with respect to v.

property SurfaceEvaluation.min_curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Minimum curvature.

property SurfaceEvaluation.min_curvature_direction:
ansys.geometry.core.math.vector.UnitVector3D

Abstractmethod

Minimum curvature direction.

property SurfaceEvaluation.max_curvature: *ansys.geometry.core.typing.Real*

Abstractmethod

Maximum curvature.

property SurfaceEvaluation.max_curvature_direction:
ansys.geometry.core.math.vector.UnitVector3D

Abstractmethod

Maximum curvature direction.

Description

Provides for evaluating a surface.

The torus.py module

Summary

Classes

<i>Torus</i>	Provides 3D torus representation.
<i>TorusEvaluation</i>	Evaluate the torus`` at given parameters.

Torus

```
class ansys.geometry.core.shapes.surfaces.torus.Torus(origin: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    ansys.geometry.core.math.point.Point3D,
    major_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real,
    minor_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real, reference:
    numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_X, axis: numpy.ndarray |
    ansys.geometry.core.typing.RealSequence |
    an-
    sys.geometry.core.math.vector.UnitVector3D |
    ansys.geometry.core.math.vector.Vector3D =
    UNITVECTOR3D_Z)
```

Bases: *ansys.geometry.core.shapes.surfaces.surface.Surface*

Provides 3D torus representation.

Parameters

origin

[[ndarray](#) | RealSequence | Point3D] Centered origin of the torus.

major_radius

[[Quantity](#) | Distance | Real] Major radius of the torus.

minor_radius

[[Quantity](#) | Distance | Real] Minor radius of the torus.

reference

[[ndarray](#) | RealSequence | UnitVector3D | Vector3D] X-axis direction.

axis

[[ndarray](#) | RealSequence | UnitVector3D | Vector3D] Z-axis direction.

Overview**Abstract methods**

<i>contains_param</i>	Check a parameter is within the parametric range of the surface.
<i>contains_point</i>	Check a point is contained by the surface.

Methods

<i>transformed_copy</i>	Create a transformed copy of the torus from a transformation matrix.
<i>mirrored_copy</i>	Create a mirrored copy of the torus along the y-axis.
<i>evaluate</i>	Evaluate the torus at the given parameters.
<i>parameterization</i>	Parameterize the torus surface as a tuple (U and V respectively).
<i>project_point</i>	Project a point onto the torus and evaluate the torus.

Properties

<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.
<i>visualization_polydata</i>	Get the visualization polydata for the torus.

Special methods

<i>__eq__</i>	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.

property `Torus.visualization_polydata`: `pyvista.PolyData`

Get the visualization polydata for the torus.

Returns

pv.PolyData

Visualization polydata for the torus.

Method detail

`Torus.__eq__(other: Torus) → bool`

Equals operator for the *Torus* class.

`Torus.transformed_copy(matrix: ansys.geometry.core.math.matrix.Matrix44) → Torus`

Create a transformed copy of the torus from a transformation matrix.

Parameters

matrix

[`Matrix44`] 4x4 transformation matrix to apply to the torus.

Returns

Torus

New torus that is the transformed copy of the original torus.

`Torus.mirrored_copy() → Torus`

Create a mirrored copy of the torus along the y-axis.

Returns

Torus

New torus that is a mirrored copy of the original torus.

Torus.evaluate(parameter: [ansys.geometry.core.shapes.parameterization.ParamUV](#)) → *TorusEvaluation*

Evaluate the torus at the given parameters.

Parameters

parameter
[ParamUV] Parameters (u,v) to evaluate the torus at.

Returns

TorusEvaluation
Resulting evaluation.

Torus.parameterization() → tuple[[ansys.geometry.core.shapes.parameterization.Parameterization](#),
[ansys.geometry.core.shapes.parameterization.Parameterization](#)]

Parameterize the torus surface as a tuple (U and V respectively).

The U parameter specifies the longitude angle, increasing clockwise (east) about the axis (right-hand corkscrew law). It has a zero parameter at `Geometry.Frame.DirX` and a period of 2π .

The V parameter specifies the latitude, increasing north, with a zero parameter at the equator. For the donut, where the major radius is greater than the minor radius, the range is $[-\pi, \pi]$ and the parameterization is periodic. For a degenerate torus, the range is restricted accordingly and the parameterization is non-periodic.

Returns

tuple[Parameterization, Parameterization]
Information about how a torus's u and v parameters are parameterized, respectively.

Torus.project_point(point: [ansys.geometry.core.math.point.Point3D](#)) → *TorusEvaluation*

Project a point onto the torus and evaluate the torus.

Parameters

point
[Point3D] Point to project onto the torus.

Returns

TorusEvaluation
Resulting evaluation.

abstractmethod **Torus.contains_param**(param_uv: [ansys.geometry.core.shapes.parameterization.ParamUV](#))
→ bool

Check a parameter is within the parametric range of the surface.

abstractmethod **Torus.contains_point**(point: [ansys.geometry.core.math.point.Point3D](#)) → bool

Check a point is contained by the surface.

The point can either lie within the surface or on its boundary.

TorusEvaluation

class [ansys.geometry.core.shapes.surfaces.torus.TorusEvaluation](#)(torus: [Torus](#), parameter: [ansys.geometry.core.shapes.parameterization.ParamU](#))

Bases: [ansys.geometry.core.shapes.surfaces.surface_evaluation.SurfaceEvaluation](#)

Evaluate the torus`` at given parameters.

Parameters

Torus: `~ansys.geometry.core.shapes-surfaces.torus.Torus`

Torust to evaluate.

parameter: `ParamUV`

Parameters (u, v) to evaluate the torus at.

Overview

Properties

<code>torus</code>	Torus being evaluated.
<code>parameter</code>	Parameter that the evaluation is based upon.
<code>position</code>	Position of the evaluation.
<code>normal</code>	Normal to the surface.
<code>u_derivative</code>	First derivative with respect to the U parameter.
<code>v_derivative</code>	First derivative with respect to the V parameter.
<code>uu_derivative</code>	Second derivative with respect to the U parameter.
<code>uv_derivative</code>	Second derivative with respect to the U and V parameters.
<code>vv_derivative</code>	Second derivative with respect to the V parameter.
<code>curvature</code>	Curvature of the torus.
<code>min_curvature</code>	Minimum curvature of the torus.
<code>min_curvature_direction</code>	Minimum curvature direction.
<code>max_curvature</code>	Maximum curvature of the torus.
<code>max_curvature_direction</code>	Maximum curvature direction.

Attributes

`cache`

Import detail

```
from ansys.geometry.core.shapes-surfaces.torus import TorusEvaluation
```

Property detail

property `TorusEvaluation.torus:` `Torus`

Torus being evaluated.

property `TorusEvaluation.parameter:` `ansys.geometry.core.shapes-parameterization.ParamUV`

Parameter that the evaluation is based upon.

property `TorusEvaluation.position:` `ansys.geometry.core.math.point.Point3D`

Position of the evaluation.

Returns

`Point3D`

Point that lies on the torus at this evaluation.

property `TorusEvaluation.normal:` `ansys.geometry.core.math.vector.UnitVector3D`

Normal to the surface.

Returns

UnitVector3D

Normal unit vector to the torus at this evaluation.

property TorusEvaluation.**u_derivative**: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the U parameter.

Returns**Vector3D**

First derivative with respect to the U parameter.

property TorusEvaluation.**v_derivative**: *ansys.geometry.core.math.vector.Vector3D*

First derivative with respect to the V parameter.

Returns**Vector3D**

First derivative with respect to the V parameter.

property TorusEvaluation.**uu_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U parameter.

Returns**Vector3D**

Second derivative with respect to the U parameter.

property TorusEvaluation.**uv_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the U and V parameters.

Returns**Vector3D**

Second derivative with respect to the U and V parameters.

property TorusEvaluation.**vv_derivative**: *ansys.geometry.core.math.vector.Vector3D*

Second derivative with respect to the V parameter.

Returns**Vector3D**

Second derivative with respect to the V parameter.

property TorusEvaluation.**curvature**: *tuple[ansys.geometry.core.typing.Real, ansys.geometry.core.math.vector.Vector3D, ansys.geometry.core.typing.Real, ansys.geometry.core.math.vector.Vector3D]*

Curvature of the torus.

Returns

tuple[Real, Vector3D, Real, Vector3D]

Minimum and maximum curvature value and direction, respectively.

property TorusEvaluation.**min_curvature**: *ansys.geometry.core.typing.Real*

Minimum curvature of the torus.

Returns**Real**

Minimum curvature of the torus.

property `TorusEvaluation.min_curvature_direction:`
`ansys.geometry.core.math.vector.UnitVector3D`

Minimum curvature direction.

Returns

UnitVector3D
Minimum curvature direction.

property `TorusEvaluation.max_curvature:` `ansys.geometry.core.typing.Real`

Maximum curvature of the torus.

Returns

Real
Maximum curvature of the torus.

property `TorusEvaluation.max_curvature_direction:`
`ansys.geometry.core.math.vector.UnitVector3D`

Maximum curvature direction.

Returns

UnitVector3D
Maximum curvature direction.

Attribute detail

`TorusEvaluation.cache`

Description

Provides for creating and managing a torus.

The `trimmed_surface.py` module

Summary

Classes

<code>TrimmedSurface</code>	Represents a trimmed surface.
<code>ReversedTrimmedSurface</code>	Represents a reversed trimmed surface.

TrimmedSurface

class `ansys.geometry.core.shapes-surfaces-trimmed_surface.TrimmedSurface`(*geometry*: `ansys.geometry.core.shapes-surfaces-surfaces`, *box_uv*: `ansys.geometry.core.shapes-box_uv.BoxUV`)

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**

<code>get_proportional_parameters</code>	Convert non-proportional parameters into proportional parameters.
<code>normal</code>	Provide the normal to the surface.
<code>project_point</code>	Project a point onto the surface and evaluate it at that location.
<code>evaluate_proportion</code>	Evaluate the surface at proportional u and v parameters.

Properties

<code>geometry</code>	Underlying mathematical surface.
<code>box_uv</code>	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-surfaces-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-trimmed_surface.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 BoxUV class.
--------------	--

Enums

<i>LocationUV</i>	Provides the enumeration for indicating locations for BoxUV.
-------------------	--

BoxUV

```
class ansys.geometry.core.shapes.box_uv.BoxUV(range_u:
                                             ansys.geometry.core.shapes.parameterization.Interval =
                                             None, range_v:
                                             ansys.geometry.core.shapes.parameterization.Interval =
                                             None)
```

Provides the implementation for BoxUV class.

Overview

Methods

<i>is_empty</i>	Check if this BoxUV is empty.
<i>proportion</i>	Evaluate the BoxUV at the given proportions.
<i>get_center</i>	Evaluate the this BoxUV in the center.
<i>is_negative</i>	Check whether the BoxUV is negative.
<i>contains</i>	Check whether the BoxUV contains a given u and v pair parameter.
<i>inflate</i>	Enlarge the BoxUV u and v intervals by deltas.
<i>get_corner</i>	Get the corner location of the BoxUV.

Properties

<i>interval_u</i>	u interval.
<i>interval_v</i>	v interval.

Special methods

<i>__eq__</i>	Check whether two BoxUV instances are equal.
<i>__ne__</i>	Check whether two BoxUV instances are not equal.

Import detail

```
from ansys.geometry.core.shapes.box_uv import BoxUV
```

Property detail

property `BoxUV.interval_u`: *ansys.geometry.core.shapes.parameterization.Interval*
u interval.

property `BoxUV.interval_v`: *ansys.geometry.core.shapes.parameterization.Interval*
v interval.

Method detail

`BoxUV.__eq__(other: object) → bool`
Check whether two BoxUV instances are equal.

`BoxUV.__ne__(other: object) → bool`
Check whether two BoxUV instances are not equal.

`BoxUV.is_empty()`
Check if this BoxUV is empty.

`BoxUV.proportion(prop_u: ansys.geometry.core.typing.Real, prop_v: ansys.geometry.core.typing.Real) → ansys.geometry.core.shapes.parameterization.ParamUV`
Evaluate the BoxUV at the given proportions.

`BoxUV.get_center()` → *ansys.geometry.core.shapes.parameterization.ParamUV*
Evaluate the this BoxUV in the center.

`BoxUV.is_negative(tolerance_u: ansys.geometry.core.typing.Real, tolerance_v: ansys.geometry.core.typing.Real) → bool`
Check whether the BoxUV is negative.

`BoxUV.contains(param: ansys.geometry.core.shapes.parameterization.ParamUV) → bool`
Check whether the BoxUV contains a given u and v pair parameter.

`BoxUV.inflate(delta_u: ansys.geometry.core.typing.Real, delta_v: ansys.geometry.core.typing.Real) → BoxUV`
Enlarge the BoxUV u and v intervals by deltas.

`BoxUV.get_corner(location: LocationUV) → ansys.geometry.core.shapes.parameterization.ParamUV`
Get the corner location of the BoxUV.

LocationUV

class `ansys.geometry.core.shapes.box_uv.LocationUV(*args, **kws)`
Bases: `enum.Enum`
Provides the enumeration for indicating locations for BoxUV.

Overview

Attributes

<i>TopLeft</i>
<i>TopCenter</i>
<i>TopRight</i>
<i>BottomLeft</i>
<i>BottomCenter</i>
<i>BottomRight</i>
<i>LeftCenter</i>
<i>RightCenter</i>
<i>Center</i>

Import detail

```
from ansys.geometry.core.shapes.box_uv import LocationUV
```

Attribute detail

`LocationUV.TopLeft = 1`

`LocationUV.TopCenter = 2`

`LocationUV.TopRight = 3`

`LocationUV.BottomLeft = 4`

`LocationUV.BottomCenter = 5`

`LocationUV.BottomRight = 6`

`LocationUV.LeftCenter = 7`

`LocationUV.RightCenter = 8`

`LocationUV.Center = 9`

Description

Provides the BoxUV class.

The parameterization.py module

Summary

Classes

<i>ParamUV</i>	Parameter class containing 2 parameters: (u, v).
<i>Interval</i>	Interval class that defines a range of values.
<i>Parameterization</i>	Parameterization class describes the parameters of a specific geometry.

Enums

<i>ParamForm</i>	ParamForm enum class that defines the form of a Parameterization.
<i>ParamType</i>	ParamType enum class that defines the type of a Parameterization.

ParamUV

class `ansys.geometry.core.shapes.parameterization.ParamUV`(*u*: `ansys.geometry.core.typing.Real`, *v*: `ansys.geometry.core.typing.Real`)

Parameter class containing 2 parameters: (u, v).

Parameters

u	[Real] U parameter.
v	[Real] V parameter.

Notes

Likened to a 2D point in UV space. Used as an argument in parametric surface evaluations. This matches the service implementation for the Geometry service.

Overview

Properties

<i>u</i>	U parameter.
<i>v</i>	V parameter.

Special methods

<code>__add__</code>	Add the u and v components of the other ParamUV to this ParamUV.
<code>__sub__</code>	Subtract the u and v components of a ParamUV from this ParamUV.
<code>__mul__</code>	Multiplies the u and v components of this ParamUV by a ParamUV.
<code>__truediv__</code>	Divides the u and v components of this ParamUV by a ParamUV.
<code>__iter__</code>	Iterate a ParamUV.
<code>__repr__</code>	Represent the ParamUV as a string.

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamUV
```

Property detail

property `ParamUV.u`: `ansys.geometry.core.typing.Real`

U parameter.

property `ParamUV.v`: `ansys.geometry.core.typing.Real`

V parameter.

Method detail

`ParamUV.__add__(other: ParamUV) → ParamUV`

Add the u and v components of the other ParamUV to this ParamUV.

Parameters**other**

[ParamUV] The parameters to add these parameters.

Returns**ParamUV**

The sum of the parameters.

`ParamUV.__sub__(other: ParamUV) → ParamUV`

Subtract the u and v components of a ParamUV from this ParamUV.

Parameters**other**

[ParamUV] The parameters to subtract from these parameters.

Returns**ParamUV**

The difference of the parameters.

`ParamUV.__mul__(other: ParamUV) → ParamUV`

Multiplies the u and v components of this ParamUV by a ParamUV.

Parameters**other**

[ParamUV] The parameters to multiply by these parameters.

Returns**ParamUV**

The product of the parameters.

`ParamUV.__truediv__(other: ParamUV) → ParamUV`

Divides the u and v components of this ParamUV by a ParamUV.

Parameters**other**

[ParamUV] The parameters to divide these parameters by.

Returns**ParamUV**

The quotient of the parameters.

`ParamUV.__iter__()`

Iterate a ParamUV.

`ParamUV.__repr__() → str`

Represent the ParamUV as a string.

Interval

class `ansys.geometry.core.shapes.parameterization.Interval`(*start*: `ansys.geometry.core.typing.Real`,
end: `ansys.geometry.core.typing.Real`)

Interval class that defines a range of values.

Parameters

start
[Real] Start value of the interval.

end
[Real] End value of the interval.

Overview

Methods

<code>is_open</code>	If the interval is open (-inf, inf).
<code>is_closed</code>	If the interval is closed. Neither value is inf or -inf.
<code>is_empty</code>	Check if the current interval is empty.
<code>get_span</code>	Get the quantity contained by the interval.
<code>get_relative_val</code>	Get an evaluation property of the interval, used in BoxUV.
<code>is_negative</code>	Indicate whether the current interval is negative.
<code>self_unite</code>	Get the union of two intervals and update the current interval.
<code>self_intersect</code>	Get the intersection of two intervals and update the current one.
<code>contains_value</code>	Check if the current interval contains the value <code>t</code> .
<code>contains</code>	Check if interval contains value <code>t</code> using default accuracy.
<code>inflate</code>	Enlarge the current interval by the given delta value.

Properties

<code>start</code>	Start value of the interval.
<code>end</code>	End value of the interval.

Attributes

<code>not_empty</code>

Static methods

<code>unite</code>	Get the union of two intervals.
<code>intersect</code>	Get the intersection of two intervals.

Special methods

<code>__eq__</code>	Compare two intervals.
<code>__repr__</code>	Represent the interval as a string.

Import detail

```
from ansys.geometry.core.shapes.parameterization import Interval
```

Property detail

property `Interval.start`: *ansys.geometry.core.typing.Real*

Start value of the interval.

property `Interval.end`: *ansys.geometry.core.typing.Real*

End value of the interval.

Attribute detail

`Interval.not_empty` = `True`

Method detail

`Interval.__eq__`(*other: object*)

Compare two intervals.

`Interval.is_open`() → `bool`

If the interval is open (-inf, inf).

Returns

`bool`

True if both ends of the interval are negative and positive infinity respectively.

`Interval.is_closed`() → `bool`

If the interval is closed. Neither value is inf or -inf.

Returns

`bool`

True if neither bound of the interval is infinite.

`Interval.is_empty`() → `bool`

Check if the current interval is empty.

Returns

`bool`

True when the interval is empty, False otherwise.

`Interval.get_span`() → *ansys.geometry.core.typing.Real*

Get the quantity contained by the interval.

The interval must be closed.

Returns

`Real`

The difference between the end and start of the interval.

`Interval.get_relative_val`(*t: ansys.geometry.core.typing.Real*) → *ansys.geometry.core.typing.Real*

Get an evaluation property of the interval, used in BoxUV.

Parameters

t
[Real] Offset to evaluate the interval at.

Returns

Real
Actual value according to the offset.

`Interval.is_negative(tolerance: ansys.geometry.core.typing.Real) → bool`

Indicate whether the current interval is negative.

Parameters

tolerance
[Real] Accepted range because the data type of the interval could be in doubles.

Returns

bool
True if the interval is negative, False otherwise.

`static Interval.unite(first: Interval, second: Interval) → Interval`

Get the union of two intervals.

Parameters

first
[Interval] First interval.

second
[Interval] Second interval.

Returns

Interval
Union of the two intervals.

`Interval.self_unite(other: Interval) → None`

Get the union of two intervals and update the current interval.

Parameters

other
[Interval] Interval to unite with.

`static Interval.intersect(first: Interval, second: Interval, tolerance: ansys.geometry.core.typing.Real) → Interval`

Get the intersection of two intervals.

Parameters

first
[Interval] First interval.

second
[Interval] Second interval.

Returns

Interval
Intersection of the two intervals.

`Interval.self_intersect(other: Interval, tolerance: ansys.geometry.core.typing.Real) → None`

Get the intersection of two intervals and update the current one.

Parameters

other

[Interval] Interval to intersect with.

tolerance

[Real] Accepted range of error given that the interval could be in float values.

`Interval.contains_value(t: ansys.geometry.core.typing.Real, accuracy: ansys.geometry.core.typing.Real) → bool`

Check if the current interval contains the value t.

Parameters

t

[Real] Value of interest.

accuracy

[Real] Accepted range of error given that the interval could be in float values.

Returns

bool

True if the interval contains the value, False otherwise.

`Interval.contains(t: ansys.geometry.core.typing.Real) → bool`

Check if interval contains value t using default accuracy.

Parameters

t

[Real] Value of interest.

Returns

bool

True if the interval contains the value, False otherwise.

`Interval.inflate(delta: ansys.geometry.core.typing.Real) → Interval`

Enlarge the current interval by the given delta value.

`Interval.__repr__() → str`

Represent the interval as a string.

Parameterization

`class ansys.geometry.core.shapes.parameterization.Parameterization(form: ParamForm, type: ParamType, interval: Interval)`

Parameterization class describes the parameters of a specific geometry.

Parameters

form

[ParamForm] Form of the parameterization.

type

[ParamType] Type of the parameterization.

interval

[Interval] Interval of the parameterization.

Overview**Properties**

<i>form</i>	The form of the parameterization.
<i>type</i>	The type of the parameterization.
<i>interval</i>	The interval of the parameterization.

Special methods

<i>__repr__</i>	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, **kws)

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, **kws)

Bases: `enum.Enum`

ParamType enum class that defines the type of a Parameterization.

Overview

Attributes

LINEAR
CIRCULAR
OTHER

Import detail

```
from ansys.geometry.core.shapes.parameterization import ParamType
```

Attribute detail

ParamType.LINEAR = 1

ParamType.CIRCULAR = 2

ParamType.OTHER = 3

Description

Provides the parametrization-related classes.

Description

Provides the PyAnsys Geometry geometry subpackage.

The sketch package

Summary

Submodules

<code>arc</code>	Provides for creating and managing an arc.
<code>box</code>	Provides for creating and managing a box (quadrilateral).
<code>circle</code>	Provides for creating and managing a circle.
<code>edge</code>	Provides for creating and managing an edge.
<code>ellipse</code>	Provides for creating and managing an ellipse.
<code>face</code>	Provides for creating and managing a face (closed 2D sketch).
<code>gears</code>	Module for creating and managing gears.
<code>nurbs</code>	Provides for creating and managing a nurbs sketch curve.
<code>polygon</code>	Provides for creating and managing a polygon.
<code>segment</code>	Provides for creating and managing a segment.
<code>sketch</code>	Provides for creating and managing a sketch.
<code>slot</code>	Provides for creating and managing a slot.
<code>trapezoid</code>	Provides for creating and managing a trapezoid.
<code>triangle</code>	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(start: ansys.geometry.core.math.point.Point2D, end:  
ansys.geometry.core.math.point.Point2D, center:  
ansys.geometry.core.math.point.Point2D, clockwise: bool =  
False)
```

Bases: `ansys.geometry.core.sketch.edge.SketchEdge`

Provides for modeling an arc.

Parameters

start

[Point2D] Starting point of the arc.

end

[Point2D] Ending point of the arc.

center

[Point2D] Center point of the arc.

clockwise

[bool, default: `False`] Whether the arc spans the clockwise angle between the start and end points. When `False` (default), the arc spans the counter-clockwise angle. When `True`, the arc spans the clockwise angle.

Overview

Constructors

<code>from_three_points</code>	Create an arc from three given points.
<code>from_start_end_and_radius</code>	Create an arc from a starting point, an ending point, and a radius.
<code>from_start_center_and_angle</code>	Create an arc from a starting point, a center point, and an angle.

Properties

<code>start</code>	Starting point of the arc line.
<code>end</code>	Ending point of the arc line.
<code>center</code>	Center point of the arc.
<code>length</code>	Length of the arc.
<code>radius</code>	Radius of the arc.
<code>angle</code>	Angle of the arc.
<code>is_clockwise</code>	Flag indicating whether the rotation of the angle is clockwise.
<code>sector_area</code>	Area of the sector of the arc.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Special methods

<code>__eq__</code>	Equals operator for the Arc class.
<code>__ne__</code>	Not equals operator for the Arc class.

Import detail

```
from ansys.geometry.core.sketch.arc import Arc
```

Property detail

property Arc.start: `ansys.geometry.core.math.point.Point2D`

Starting point of the arc line.

property Arc.end: `ansys.geometry.core.math.point.Point2D`

Ending point of the arc line.

property Arc.center: `ansys.geometry.core.math.point.Point2D`

Center point of the arc.

property Arc.length: `pint.Quantity`

Length of the arc.

property Arc.radius: `pint.Quantity`

Radius of the arc.

property Arc.angle: `pint.Quantity`

Angle of the arc.

property Arc.is_clockwise: `bool`

Flag indicating whether the rotation of the angle is clockwise.

Returns**bool**

True if the sense of rotation is clockwise. False if the sense of rotation is counter-clockwise.

property `Arc.sector_area`: **pint.Quantity**

Area of the sector of the arc.

property `Arc.visualization_polydata`: **pyvista.PolyData**

VTK polydata representation for PyVista visualization.

Returns**pyvista.PolyData**

VTK pyvista.Polydata configuration.

Notes

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Method detail`Arc.__eq__(other: Arc) → bool`Equals operator for the `Arc` class.`Arc.__ne__(other: Arc) → bool`Not equals operator for the `Arc` class.**classmethod** `Arc.from_three_points`(*start*: `ansys.geometry.core.math.point.Point2D`, *inter*: `ansys.geometry.core.math.point.Point2D`, *end*: `ansys.geometry.core.math.point.Point2D`)

Create an arc from three given points.

Parameters**start**[`Point2D`] Starting point of the arc.**inter**[`Point2D`] Intermediate point (location) of the arc.**end**[`Point2D`] Ending point of the arc.**Returns****Arc**

Arc generated from the three points.

classmethod `Arc.from_start_end_and_radius`(*start*: `ansys.geometry.core.math.point.Point2D`, *end*: `ansys.geometry.core.math.point.Point2D`, *radius*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance` | `ansys.geometry.core.typing.Real`, *convex_arc*: *bool* = *False*, *clockwise*: *bool* = *False*)

Create an arc from a starting point, an ending point, and a radius.

Parameters**start**[`Point2D`] Starting point of the arc.

end

[Point2D] Ending point of the arc.

radius

[Quantity | Distance | Real] Radius of the arc.

convex_arc[bool, default: `False`] Whether the arc is convex. The default is `False`. When `False`, the arc is concave. When `True`, the arc is convex.**clockwise**[bool, default: `False`] Whether the arc spans the clockwise angle between the start and end points. When `False`, the arc spans the counter-clockwise angle. When `True`, the arc spans the clockwise angle.**Returns****Arc**

Arc generated from the three points.

classmethod `Arc.from_start_center_and_angle`(*start*: `ansys.geometry.core.math.point.Point2D`, *center*: `ansys.geometry.core.math.point.Point2D`, *angle*: `ansys.geometry.core.misc.measurements.Angle` | `pint.Quantity` | `ansys.geometry.core.typing.Real`, *clockwise*: `bool` = `False`)

Create an arc from a starting point, a center point, and an angle.

Parameters**start**

[Point2D] Starting point of the arc.

center

[Point2D] Center point of the arc.

angle

[Angle | Quantity | Real] Angle of the arc.

clockwise[bool, default: `False`] Whether the provided angle should be considered clockwise. When `False`, the angle is considered counter-clockwise. When `True`, the angle is considered clockwise.**Returns****Arc**

Arc generated from the three points.

Description

Provides for creating and managing an arc.

The `box.py` module**Summary****Classes**

<i>Box</i>	Provides for modeling a box.
------------	------------------------------

Box

```
class ansys.geometry.core.sketch.box.Box(center: ansys.geometry.core.math.point.Point2D, width: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, height: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0)
```

Bases: *ansys.geometry.core.sketch.face.SketchFace*

Provides for modeling a box.

Parameters

center: Point2D

Center point of the box.

width

[Quantity | Distance | Real] Width of the box.

height

[Quantity | Distance | Real] Height of the box.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Overview

Properties

<i>center</i>	Center point of the box.
<i>width</i>	Width of the box.
<i>height</i>	Height of the box.
<i>perimeter</i>	Perimeter of the box.
<i>area</i>	Area of the box.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.box import Box
```

Property detail

property Box.**center:** *ansys.geometry.core.math.point.Point2D*

Center point of the box.

property Box.**width:** *pint.Quantity*

Width of the box.

property Box.**height:** *pint.Quantity*

Height of the box.

property `Box.perimeter: pint.Quantity`

Perimeter of the box.

property `Box.area: pint.Quantity`

Area of the box.

property `Box.visualization_polydata: pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a box (quadrilateral).

The circle.py module

Summary

Classes

<i>SketchCircle</i>	Provides for modeling a circle.
---------------------	---------------------------------

SketchCircle

class `ansys.geometry.core.sketch.circle.SketchCircle`(*center*: `ansys.geometry.core.math.point.Point2D`,
radius: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Distance`
| `ansys.geometry.core.typing.Real`, *plane*:
`ansys.geometry.core.math.plane.Plane` =
Plane())

Bases: `ansys.geometry.core.sketch.face.SketchFace`, `ansys.geometry.core.shapes.curves.circle.Circle`

Provides for modeling a circle.

Parameters

center: Point2D

Center point of the circle.

radius

[`Quantity` | `Distance` | `Real`] Radius of the circle.

plane

[`Plane`, optional] Plane containing the sketched circle, which is the global XY plane by default.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing the SketchCircle objects.
---------------------------	---

Properties

<code>center</code>	Center of the circle.
<code>perimeter</code>	Perimeter of the circle.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.circle import SketchCircle
```

Property detail

property SketchCircle.`center`: `ansys.geometry.core.math.point.Point2D`

Center of the circle.

property SketchCircle.`perimeter`: `pint.Quantity`

Perimeter of the circle.

Notes

This property resolves the dilemma between using the `SkethFace.perimeter` property and the `Circle.perimeter` property.

property SketchCircle.`visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Method detail

SketchCircle.`plane_change`(*plane*: `ansys.geometry.core.math.plane.Plane`) → None

Redefine the plane containing the SketchCircle objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched circle.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing a circle.

The `edge.py` module

Summary

Classes

<i>SketchEdge</i>	Provides for modeling edges forming sketched shapes.
-------------------	--

SketchEdge

class `ansys.geometry.core.sketch.edge.SketchEdge`

Provides for modeling edges forming sketched shapes.

Overview

Abstract methods

<i>contains_point</i>	Check if the edge contains the given point within a tolerance.
-----------------------	--

Methods

<i>plane_change</i>	Redefine the plane containing SketchEdge objects.
---------------------	---

Properties

<i>start</i>	Starting point of the edge.
<i>end</i>	Ending point of the edge.
<i>length</i>	Length of the edge.
<i>visualization_polydata</i>	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

abstractmethod `SketchEdge.contains_point`(*point*: `ansys.geometry.core.math.point.Point2D`, *tol*: *float* = *1e-06*) → `bool`

Check if the edge contains the given point within a tolerance.

`SketchEdge.plane_change`(*plane*: `ansys.geometry.core.math.plane.Plane`) → `None`

Redefine the plane containing `SketchEdge` objects.

Parameters

plane

[`Plane`] Desired new plane that is to contain the sketched edge.

Notes

This implies that their 3D definition might suffer changes. By default, this method does nothing. It is required to be implemented in child `SketchEdge` classes.

Description

Provides for creating and managing an edge.

The `ellipse.py` module

Summary

Classes

<code>SketchEllipse</code>	Provides for modeling an ellipse.
----------------------------	-----------------------------------

`SketchEllipse`

```
class ansys.geometry.core.sketch.ellipse.SketchEllipse(center:
    ansys.geometry.core.math.point.Point2D,
    major_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real,
    minor_radius: pint.Quantity | an-
    sys.geometry.core.misc.measurements.Distance
    | ansys.geometry.core.typing.Real, angle:
    pint.Quantity | an-
    sys.geometry.core.misc.measurements.Angle
    | ansys.geometry.core.typing.Real = 0,
    plane:
    ansys.geometry.core.math.plane.Plane =
    Plane())
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`, `ansys.geometry.core.shapes.curves.ellipse.Ellipse`

Provides for modeling an ellipse.

Parameters

center: Point2D

Center point of the ellipse.

major_radius

[Quantity | Distance | Real] Major radius of the ellipse.

minor_radius

[Quantity | Distance | Real] Minor radius of the ellipse.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

plane

[Plane, optional] Plane containing the sketched ellipse, which is the global XY plane by default.

Overview

Methods

<code>plane_change</code>	Redefine the plane containing SketchEllipse objects.
---------------------------	--

Properties

<code>center</code>	Center point of the ellipse.
<code>angle</code>	Orientation angle of the ellipse.
<code>perimeter</code>	Perimeter of the circle.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.ellipse import SketchEllipse
```

Property detail

property SketchEllipse.center: *ansys.geometry.core.math.point.Point2D*

Center point of the ellipse.

property SketchEllipse.angle: *pint.Quantity*

Orientation angle of the ellipse.

property SketchEllipse.perimeter: *pint.Quantity*

Perimeter of the circle.

Notes

This property resolves the dilemma between using the *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

<code>plane_change</code>	Redefine the plane containing SketchFace objects.
---------------------------	---

Properties

<code>edges</code>	List of all component edges forming the face.
<code>perimeter</code>	Perimeter of the face.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.face import SketchFace
```

Property detail

property `SketchFace.edges`: `list[ansys.geometry.core.sketch.edge.SketchEdge]`

List of all component edges forming the face.

property `SketchFace.perimeter`: `pint.Quantity`

Perimeter of the face.

property `SketchFace.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Method detail

`SketchFace.plane_change(plane: ansys.geometry.core.math.plane.Plane) → None`

Redefine the plane containing SketchFace objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched face.

Notes

This implies that their 3D definition might suffer changes. This method does nothing by default. It is required to be implemented in child `SketchFace` classes.

Description

Provides for creating and managing a face (closed 2D sketch).

The `gears.py` module

Summary

Classes

<code>Gear</code>	Provides the base class for sketching gears.
<code>DummyGear</code>	Provides the dummy class for sketching gears.
<code>SpurGear</code>	Provides the class for sketching spur gears.

Gear

class `ansys.geometry.core.sketch.gears.Gear`

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides the base class for sketching gears.

Overview

Properties

<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.
-------------------------------------	--

Import detail

```
from ansys.geometry.core.sketch.gears import Gear
```

Property detail

property `Gear.visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK `pyvista.Polydata` configuration.

DummyGear

```
class ansys.geometry.core.sketch.gears.DummyGear(origin: ansys.geometry.core.math.point.Point2D,
                                                  outer_radius: pint.Quantity |
                                                  ansys.geometry.core.misc.measurements.Distance |
                                                  ansys.geometry.core.typing.Real, inner_radius:
                                                  pint.Quantity |
                                                  ansys.geometry.core.misc.measurements.Distance |
                                                  ansys.geometry.core.typing.Real, n_teeth: int)
```

Bases: Gear

Provides the dummy class for sketching gears.

Parameters

origin
[Point2D] Origin of the gear.

outer_radius
[Quantity | Distance | Real] Outer radius of the gear.

inner_radius
[Quantity | Distance | Real] Inner radius of the gear.

n_teeth
[int] Number of teeth of the gear.

Import detail

```
from ansys.geometry.core.sketch.gears import DummyGear
```

SpurGear

```
class ansys.geometry.core.sketch.gears.SpurGear(origin: ansys.geometry.core.math.point.Point2D,
                                                  module: ansys.geometry.core.typing.Real,
                                                  pressure_angle: pint.Quantity |
                                                  ansys.geometry.core.misc.measurements.Angle |
                                                  ansys.geometry.core.typing.Real, n_teeth: int)
```

Bases: Gear

Provides the class for sketching spur gears.

Parameters

origin
[Point2D] Origin of the spur gear.

module
[Real] Module of the spur gear. This is also the ratio between the pitch circle diameter in millimeters and the number of teeth.

pressure_angle
[Quantity | Angle | Real] Pressure angle of the spur gear.

n_teeth
[int] Number of teeth of the spur gear.

Overview

Properties

<i>origin</i>	Origin of the spur gear.
<i>module</i>	Module of the spur gear.
<i>pressure_angle</i>	Pressure angle of the spur gear.
<i>n_teeth</i>	Number of teeth of the spur gear.
<i>ref_diameter</i>	Reference diameter of the spur gear.
<i>base_diameter</i>	Base diameter of the spur gear.
<i>addendum</i>	Addendum of the spur gear.
<i>dedendum</i>	Dedendum of the spur gear.
<i>tip_diameter</i>	Tip diameter of the spur gear.
<i>root_diameter</i>	Root diameter of the spur gear.

Import detail

```
from ansys.geometry.core.sketch.gears import SpurGear
```

Property detail

property `SpurGear.origin`: *ansys.geometry.core.math.point.Point2D*

Origin of the spur gear.

property `SpurGear.module`: *ansys.geometry.core.typing.Real*

Module of the spur gear.

property `SpurGear.pressure_angle`: *pint.Quantity*

Pressure angle of the spur gear.

property `SpurGear.n_teeth`: *int*

Number of teeth of the spur gear.

property `SpurGear.ref_diameter`: *ansys.geometry.core.typing.Real*

Reference diameter of the spur gear.

property `SpurGear.base_diameter`: *ansys.geometry.core.typing.Real*

Base diameter of the spur gear.

property `SpurGear.addendum`: *ansys.geometry.core.typing.Real*

Addendum of the spur gear.

property `SpurGear.dedendum`: *ansys.geometry.core.typing.Real*

Dedendum of the spur gear.

property `SpurGear.tip_diameter`: *ansys.geometry.core.typing.Real*

Tip diameter of the spur gear.

property `SpurGear.root_diameter`: *ansys.geometry.core.typing.Real*

Root diameter of the spur gear.

Description

Module for creating and managing gears.

The `nurbs.py` module

Summary

Classes

<i>SketchNurbs</i>	Represents a NURBS sketch curve.
--------------------	----------------------------------

SketchNurbs

class `ansys.geometry.core.sketch.nurbs.SketchNurbs`

Bases: `ansys.geometry.core.sketch.edge.SketchEdge`

Represents a NURBS sketch curve.

Warning

NURBS sketching is only supported in 26R1 and later versions of Ansys.

Notes

This class is a wrapper around the NURBS curve class from the *geomdl* library. By leveraging the *geomdl* library, this class provides a high-level interface to create and manipulate NURBS curves. The *geomdl* library is a powerful library for working with NURBS curves and surfaces. For more information, see <https://pypi.org/project/geomdl/>.

Overview

Constructors

<i>fit_curve_from_points</i>	Fit a NURBS curve to a set of points.
------------------------------	---------------------------------------

Methods

<i>contains_point</i>	Check if the curve contains a given point within a specified tolerance.
-----------------------	---

Properties

<i>geomdl_nurbs_curve</i>	Get the underlying NURBS curve.
<i>control_points</i>	Get the control points of the curve.
<i>degree</i>	Get the degree of the curve.
<i>knots</i>	Get the knot vector of the curve.
<i>weights</i>	Get the weights of the control points.
<i>start</i>	Get the start point of the curve.
<i>end</i>	Get the end point of the curve.
<i>visualization_polydata</i>	Get the VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.nurbs import SketchNurbs
```

Property detail

property `SketchNurbs.geomdl_nurbs_curve: geomdl.NURBS.Curve`

Get the underlying NURBS curve.

Notes

This property gives access to the full functionality of the NURBS curve coming from the *geomdl* library. Use with caution.

property `SketchNurbs.control_points: list[ansys.geometry.core.math.point.Point2D]`

Get the control points of the curve.

property `SketchNurbs.degree: int`

Get the degree of the curve.

property `SketchNurbs.knots: list[ansys.geometry.core.typing.Real]`

Get the knot vector of the curve.

property `SketchNurbs.weights: list[ansys.geometry.core.typing.Real]`

Get the weights of the control points.

property `SketchNurbs.start: ansys.geometry.core.math.point.Point2D`

Get the start point of the curve.

property `SketchNurbs.end: ansys.geometry.core.math.point.Point2D`

Get the end point of the curve.

property `SketchNurbs.visualization_polydata: pyvista.PolyData`

Get the 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

`SketchNurbs.contains_point(point: ansys.geometry.core.math.point.Point2D, tolerance: ansys.geometry.core.typing.Real = 1e-06) → bool`

Check if the curve contains a given point within a specified tolerance.

Parameters

point

[Point2D] The point to check.

tolerance

[Real, optional] The tolerance for the containment check, by default 1e-6.

Returns

bool

True if the curve contains the point within the tolerance, False otherwise.

classmethod `SketchNurbs.fit_curve_from_points`(*points*: *list*[*ansys.geometry.core.math.point.Point2D*],
degree: *int* = 3) → *SketchNurbs*

Fit a NURBS curve to a set of points.

Parameters

points

[*list*[*Point2D*]] The points to fit the curve to.

degree

[*int*, optional] The degree of the NURBS curve, by default 3.

Returns

SketchNurbs

A new instance of SketchNurbs fitted to the given points.

Description

Provides for creating and managing a nurbs sketch curve.

The polygon.py module

Summary

Classes

<i>Polygon</i>	Provides for modeling regular polygons.
----------------	---

Polygon

class `ansys.geometry.core.sketch.polygon.Polygon`(*center*: *ansys.geometry.core.math.point.Point2D*,
inner_radius: *pint.Quantity* |
ansys.geometry.core.misc.measurements.Distance |
ansys.geometry.core.typing.Real, *sides*: *int*, *angle*:
pint.Quantity |
ansys.geometry.core.misc.measurements.Angle |
ansys.geometry.core.typing.Real = 0)

Bases: *ansys.geometry.core.sketch.face.SketchFace*

Provides for modeling regular polygons.

Parameters

center: Point2D

Center point of the circle.

inner_radius

[*Quantity* | *Distance* | *Real*] Inner radius (apothem) of the polygon.

sides

[*int*] Number of sides of the polygon.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Overview**Properties**

<code>center</code>	Center point of the polygon.
<code>inner_radius</code>	Inner radius (apothem) of the polygon.
<code>n_sides</code>	Number of sides of the polygon.
<code>angle</code>	Orientation angle of the polygon.
<code>length</code>	Side length of the polygon.
<code>outer_radius</code>	Outer radius of the polygon.
<code>perimeter</code>	Perimeter of the polygon.
<code>area</code>	Area of the polygon.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.polygon import Polygon
```

Property detail

property Polygon.`center`: `ansys.geometry.core.math.point.Point2D`

Center point of the polygon.

property Polygon.`inner_radius`: `pint.Quantity`

Inner radius (apothem) of the polygon.

property Polygon.`n_sides`: `int`

Number of sides of the polygon.

property Polygon.`angle`: `pint.Quantity`

Orientation angle of the polygon.

property Polygon.`length`: `pint.Quantity`

Side length of the polygon.

property Polygon.`outer_radius`: `pint.Quantity`

Outer radius of the polygon.

property Polygon.`perimeter`: `pint.Quantity`

Perimeter of the polygon.

property Polygon.`area`: `pint.Quantity`

Area of the polygon.

property Polygon.`visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a polygon.

The `segment.py` module

Summary

Classes

<i>SketchSegment</i>	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

<i>plane_change</i>	Redefine the plane containing SketchSegment objects.
---------------------	--

Properties

<i>start</i>	Starting point of the segment.
<i>end</i>	Ending point of the segment.
<i>length</i>	Length of the segment.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Special methods

<code>__eq__</code>	Equals operator for the SketchSegment class.
<code>__ne__</code>	Not equals operator for the SketchSegment class.

Import detail

```
from ansys.geometry.core.sketch.segment import SketchSegment
```

Property detail

property SketchSegment.start: *ansys.geometry.core.math.point.Point2D*

Starting point of the segment.

property SketchSegment.end: *ansys.geometry.core.math.point.Point2D*

Ending point of the segment.

property SketchSegment.length: *pint.Quantity*

Length of the segment.

property SketchSegment.visualization_polydata: *pyvista.PolyData*

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData

VTK pyvista.Polydata configuration.

Method detail

SketchSegment.__eq__(other: SketchSegment) → bool

Equals operator for the SketchSegment class.

SketchSegment.__ne__(other: SketchSegment) → bool

Not equals operator for the SketchSegment class.

SketchSegment.plane_change(plane: ansys.geometry.core.math.plane.Plane) → None

Redefine the plane containing SketchSegment objects.

Parameters

plane

[Plane] Desired new plane that is to contain the sketched segment.

Notes

This implies that their 3D definition might suffer changes.

Description

Provides for creating and managing a segment.

The sketch.py module

Summary

Classes

<i>Sketch</i>	Provides for building 2D sketch elements.
---------------	---

Attributes

<i>SketchObject</i>	Type to refer to both <i>SketchEdge</i> and <i>SketchFace</i> .
---------------------	---

Sketch

class ansys.geometry.core.sketch.sketch.**Sketch**(plane: ansys.geometry.core.math.plane.Plane = Plane())

Provides for building 2D sketch elements.

Overview

Methods

<i>translate_sketch_plane</i>	Translate the origin location of the active sketch plane.
<i>translate_sketch_plane_by_offset</i>	Translate the origin location of the active sketch plane by offsets.
<i>translate_sketch_plane_by_distance</i>	Translate the origin location active sketch plane by distance.
<i>get</i>	Get a list of shapes with a given tag.
<i>face</i>	Add a sketch face to the sketch.
<i>edge</i>	Add a sketch edge to the sketch.
<i>select</i>	Add all objects that match provided tags to the current context.
<i>segment</i>	Add a segment sketch object to the sketch plane.
<i>segment_to_point</i>	Add a segment to the sketch plane starting from the previous end point.
<i>segment_from_point_and_vector</i>	Add a segment to the sketch starting from a given starting point.
<i>segment_from_vector</i>	Add a segment to the sketch starting from the previous end point.
<i>arc</i>	Add an arc to the sketch plane.
<i>arc_to_point</i>	Add an arc to the sketch starting from the previous end point.
<i>arc_from_three_points</i>	Add an arc to the sketch plane from three given points.
<i>arc_from_start_end_and_radius</i>	Add an arc from the start, end points and a radius.
<i>arc_from_start_center_and_angle</i>	Add an arc from the start, center point, and angle.
<i>nurbs_from_2d_points</i>	Add a NURBS curve from a list of 2D points.
<i>triangle</i>	Add a triangle to the sketch using given vertex points.
<i>trapezoid</i>	Add a trapezoid 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.

continues on next page

Table 5 – continued from previous page

<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.
<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:` `list[ansys.geometry.core.sketch.edge.SketchEdge]`

List of all independently sketched edges.

Notes

Independently sketched edges are not assigned to a face. Face edges are not included in this list.

property `Sketch.faces:` `list[ansys.geometry.core.sketch.face.SketchFace]`

List of all independently sketched faces.

Method detail

`Sketch.translate_sketch_plane(translation: ansys.geometry.core.math.vector.Vector3D) → Sketch`

Translate the origin location of the active sketch plane.

Parameters

translation

[Vector3D] Vector defining the translation. Default units are the expected units, otherwise it will be inconsistent.

Returns

Sketch

Revised sketch state ready for further sketch actions.

```
Sketch.translate_sketch_plane_by_offset(x: pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance = Quantity(0,
    DEFAULT_UNITS.LENGTH), y: pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance = Quantity(0,
    DEFAULT_UNITS.LENGTH), z: pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance = Quantity(0,
    DEFAULT_UNITS.LENGTH)) → Sketch
```

Translate the origin location of the active sketch plane by offsets.

Parameters

x

[Quantity | Distance, default: Quantity(0, DEFAULT_UNITS.LENGTH)] Amount to translate the origin of the x-direction.

y

[Quantity | Distance, default: Quantity(0, DEFAULT_UNITS.LENGTH)] Amount to translate the origin of the y-direction.

z

[Quantity | Distance, default: Quantity(0, DEFAULT_UNITS.LENGTH)] Amount to translate the origin of the z-direction.

Returns

Sketch

Revised sketch state ready for further sketch actions.

```
Sketch.translate_sketch_plane_by_distance(direction: ansys.geometry.core.math.vector.UnitVector3D,
    distance: pint.Quantity |
    ansys.geometry.core.misc.measurements.Distance) → Sketch
```

Translate the origin location active sketch plane by distance.

Parameters

direction

[UnitVector3D] Direction to translate the origin.

distance

[Quantity | Distance] Distance to translate the origin.

Returns

Sketch

Revised sketch state ready for further sketch actions.

```
Sketch.get(tag: str) → list[SketchObject]
```

Get a list of shapes with a given tag.

Parameters

tag

[str] Tag to query against.

```
Sketch.face(face: ansys.geometry.core.sketch.face.SketchFace, tag: str | None = None) → Sketch
```

Add a sketch face to the sketch.

Parameters

face

[SketchFace] Face to add.

tag

[str, default: None] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Sketch.edge(edge: [ansys.geometry.core.sketch.edge.SketchEdge](#), tag: str | None = None) → *Sketch*

Add a sketch edge to the sketch.

Parameters

edge

[SketchEdge] Edge to add.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Sketch.select(*tags: str) → *Sketch*

Add all objects that match provided tags to the current context.

Sketch.segment(start: [ansys.geometry.core.math.point.Point2D](#), end: [ansys.geometry.core.math.point.Point2D](#), tag: str | None = None) → *Sketch*

Add a segment sketch object to the sketch plane.

Parameters

start

[Point2D] Starting point of the line segment.

end

[Point2D] Ending point of the line segment.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Sketch.segment_to_point(end: [ansys.geometry.core.math.point.Point2D](#), tag: str | None = None) → *Sketch*

Add a segment to the sketch plane starting from the previous end point.

Parameters

end

[Point2D] Ending point of the line segment.

tag

[str, default: None] User-defined label for identifying the edge.

Returns

Sketch

Revised sketch state ready for further sketch actions.

Notes

The starting point of the created edge is based upon the current context of the sketch, such as the end point of a previously added edge.

`Sketch.segment_from_point_and_vector(start: ansys.geometry.core.math.point.Point2D, vector: ansys.geometry.core.math.vector.Vector2D, tag: str | None = None)`

Add a segment to the sketch starting from a given starting point.

Parameters**start**

[Point2D] Starting point of the line segment.

vector

[Vector2D] Vector defining the line segment. Vector magnitude determines the segment endpoint. Vector magnitude is assumed to be in the same unit as the starting point.

tag

[str, default: None] User-defined label for identifying the edge.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Notes

Vector magnitude determines the segment endpoint. Vector magnitude is assumed to use the same unit as the starting point.

`Sketch.segment_from_vector(vector: ansys.geometry.core.math.vector.Vector2D, tag: str | None = None)`

Add a segment to the sketch starting from the previous end point.

Parameters**vector**

[Vector2D] Vector defining the line segment.

tag

[str, default: None] User-defined label for identifying the edge.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Notes

The starting point of the created edge is based upon the current context of the sketch, such as the end point of a previously added edge.

Vector magnitude determines the segment endpoint. Vector magnitude is assumed to use the same unit as the starting point in the previous context.

`Sketch.arc(start: ansys.geometry.core.math.point.Point2D, end: ansys.geometry.core.math.point.Point2D, center: ansys.geometry.core.math.point.Point2D, clockwise: bool = False, tag: str | None = None) → Sketch`

Add an arc to the sketch plane.

Parameters**start**

[Point2D] Starting point of the arc.

end

[Point2D] Ending point of the arc.

center

[Point2D] Center point of the arc.

clockwise

[bool, default: `False`] Whether the arc spans the angle clockwise between the start and end points. When `False` (default), the arc spans the angle counter-clockwise. When `True`, the arc spans the angle clockwise.

tag

[str, default: `None`] User-defined label for identifying the edge.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

`Sketch.arc_to_point(end: ansys.geometry.core.math.point.Point2D, center: ansys.geometry.core.math.point.Point2D, clockwise: bool = False, tag: str | None = None) → Sketch`

Add an arc to the sketch starting from the previous end point.

Parameters**end**

[Point2D] Ending point of the arc.

center

[Point2D] Center point of the arc.

clockwise

[bool, default: `False`] Whether the arc spans the angle clockwise between the start and end points. When `False` (default), the arc spans the angle counter-clockwise. When `True`, the arc spans the angle clockwise.

tag

[str, default: `None`] User-defined label for identifying the edge.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Notes

The starting point of the created edge is based upon the current context of the sketch, such as the end point of a previously added edge.

`Sketch.arc_from_three_points(start: ansys.geometry.core.math.point.Point2D, inter: ansys.geometry.core.math.point.Point2D, end: ansys.geometry.core.math.point.Point2D, tag: str | None = None) → Sketch`

Add an arc to the sketch plane from three given points.

Parameters

start
[Point2D] Starting point of the arc.

inter
[Point2D] Intermediate point (location) of the arc.

end
[Point2D] End point of the arc.

tag
[str, default: None] User-defined label for identifying the edge.

Returns

Sketch
Revised sketch state ready for further sketch actions.

`Sketch.arc_from_start_end_and_radius(start: ansys.geometry.core.math.point.Point2D, end: ansys.geometry.core.math.point.Point2D, radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, convex_arc: bool = False, clockwise: bool = False, tag: str | None = None) → Sketch`

Add an arc from the start, end points and a radius.

Parameters

start
[Point2D] Starting point of the arc.

end
[Point2D] Ending point of the arc.

radius
[Quantity | Distance | Real] Radius of the arc.

convex_arc
[bool, default: False] Whether the arc is convex. The default is False. When False, the arc spans the concave version of the arc. When True, the arc spans the convex version of the arc.

clockwise
[bool, default: False] Whether the arc spans the angle clockwise between the start and end points. When False, the arc spans the angle counter-clockwise. When True, the arc spans the angle clockwise.

tag
[str, default: None] User-defined label for identifying the edge.

Returns

Sketch
Revised sketch state ready for further sketch actions.

Warning

This method may not produce the exact same radius, depending on the input parameters. This is due to the mathematical approximations involved in the arc creation.

`Sketch.arc_from_start_center_and_angle`(*start*: `ansys.geometry.core.math.point.Point2D`, *center*: `ansys.geometry.core.math.point.Point2D`, *angle*: `pint.Quantity` | `ansys.geometry.core.misc.measurements.Angle` | `ansys.geometry.core.typing.Real`, *clockwise*: `bool` = `False`, *tag*: `str` | `None` = `None`) → `Sketch`

Add an arc from the start, center point, and angle.

Parameters

start

[`Point2D`] Starting point of the arc.

center

[`Point2D`] Center point of the arc.

angle

[`Quantity` | `Angle` | `Real`] Angle of the arc.

clockwise

[`bool`, default: `False`] Whether the arc spans the angle clockwise. The default is `False`. When `False`, the arc spans the angle counter-clockwise. When `True`, the arc spans the angle clockwise.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.nurbs_from_2d_points`(*points*: `list[ansys.geometry.core.math.point.Point2D]`, *tag*: `str` | `None` = `None`) → `Sketch`

Add a NURBS curve from a list of 2D points.

Parameters

points

[`list[Point2D]`] List of 2D points to define the NURBS curve.

tag

[`str` | `None`, default: `None`] User-defined label for identifying the curve.

Returns

Sketch

Revised sketch state ready for further sketch actions.

`Sketch.triangle`(*point1*: `ansys.geometry.core.math.point.Point2D`, *point2*: `ansys.geometry.core.math.point.Point2D`, *point3*: `ansys.geometry.core.math.point.Point2D`, *tag*: `str` | `None` = `None`) → `Sketch`

Add a triangle to the sketch using given vertex points.

Parameters

point1

[`Point2D`] Point that represents a vertex of the triangle.

point2

[`Point2D`] Point that represents a vertex of the triangle.

point3

[`Point2D`] Point that represents a vertex of the triangle.

tag

[`str`, default: `None`] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Sketch.trapezoid(*base_width*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*, *height*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*, *base_angle*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Angle* | *ansys.geometry.core.typing.Real*, *base_asymmetric_angle*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Angle* | *ansys.geometry.core.typing.Real* | *None* = *None*, *center*: *ansys.geometry.core.math.point.Point2D* = *ZERO_POINT2D*, *angle*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Angle* | *ansys.geometry.core.typing.Real* = 0, *tag*: *str* | *None* = *None*) → *Sketch*

Add a trapezoid to the sketch using given vertex points.

Parameters**base_width**

[*Quantity* | *Distance* | *Real*] Width of the lower base of the trapezoid.

height

[*Quantity* | *Distance* | *Real*] Height of the slot.

base_angle

[*Quantity* | *Distance* | *Real*] Angle for trapezoid generation. Represents the angle on the base of the trapezoid.

base_asymmetric_angle

[*Quantity* | *Angle* | *Real* | *None*, default: *None*] Asymmetrical angles on each side of the trapezoid. The default is *None*, in which case the trapezoid is symmetrical. If provided, the trapezoid is asymmetrical and the right corner angle at the base of the trapezoid is set to the provided value.

center: Point2D, default: ZERO_POINT2D

Center point of the trapezoid.

angle

[*Quantity* | *Angle* | *Real*, default: 0] Placement angle for orientation alignment.

tag

[*str*, default: *None*] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Notes

If an asymmetric base angle is defined, the base angle is applied to the left-most angle, and the asymmetric base angle is applied to the right-most angle.

Sketch.circle(*center*: *ansys.geometry.core.math.point.Point2D*, *radius*: *pint.Quantity* | *ansys.geometry.core.misc.measurements.Distance* | *ansys.geometry.core.typing.Real*, *tag*: *str* | *None* = *None*) → *Sketch*

Add a circle to the plane at a given center.

Parameters

center: Point2D

Center point of the circle.

radius

[Quantity | Distance | Real] Radius of the circle.

tag

[str, default: None] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Sketch.box(center: ansys.geometry.core.math.point.Point2D, width: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, height: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0, tag: str | None = None) → Sketch

Create a box on the sketch.

Parameters**center: Point2D**

Center point of the box.

width

[Quantity | Distance | Real] Width of the box.

height

[Quantity | Distance | Real] Height of the box.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

tag

[str, default: None] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

Sketch.slot(center: ansys.geometry.core.math.point.Point2D, width: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, height: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0, tag: str | None = None) → Sketch

Create a slot on the sketch.

Parameters**center: Point2D**

Center point of the slot.

width

[Quantity | Distance | Real] Width of the slot.

height

[Quantity | Distance | Real] Height of the slot.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

tag
 [str, default: None] User-defined label for identifying the face.

Returns

Sketch
 Revised sketch state ready for further sketch actions.

Sketch.ellipse(center: ansys.geometry.core.math.point.Point2D, major_radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, minor_radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0, tag: str | None = None) → Sketch

Create an ellipse on the sketch.

Parameters

center: Point2D
 Center point of the ellipse.

major_radius
 [Quantity | Distance | Real] Semi-major axis of the ellipse.

minor_radius
 [Quantity | Distance | Real] Semi-minor axis of the ellipse.

angle
 [Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

tag
 [str, default: None] User-defined label for identifying the face.

Returns

Sketch
 Revised sketch state ready for further sketch actions.

Sketch.polygon(center: ansys.geometry.core.math.point.Point2D, inner_radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, sides: int, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0, tag: str | None = None) → Sketch

Create a polygon on the sketch.

Parameters

center: Point2D
 Center point of the polygon.

inner_radius
 [Quantity | Distance | Real] Inner radius (apothem) of the polygon.

sides
 [int] Number of sides of the polygon.

angle
 [Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

tag
 [str, default: None] User-defined label for identifying the face.

Returns

Sketch

Revised sketch state ready for further sketch actions.

```
Sketch.dummy_gear(origin: ansys.geometry.core.math.point.Point2D, outer_radius: pint.Quantity |  
    ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real,  
    inner_radius: pint.Quantity | ansys.geometry.core.misc.measurements.Distance |  
    ansys.geometry.core.typing.Real, n_teeth: int, tag: str | None = None) → Sketch
```

Create a dummy gear on the sketch.

Parameters**origin**

[Point2D] Origin of the gear.

outer_radius

[Quantity | Distance | Real] Outer radius of the gear.

inner_radius

[Quantity | Distance | Real] Inner radius of the gear.

n_teeth

[int] Number of teeth of the gear.

tag

[str, default: None] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

```
Sketch.spur_gear(origin: ansys.geometry.core.math.point.Point2D, module: ansys.geometry.core.typing.Real,  
    pressure_angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle |  
    ansys.geometry.core.typing.Real, n_teeth: int, tag: str | None = None) → Sketch
```

Create a spur gear on the sketch.

Parameters**origin**

[Point2D] Origin of the spur gear.

module

[Real] Module of the spur gear. This is also the ratio between the pitch circle diameter in millimeters and the number of teeth.

pressure_angle

[Quantity | Angle | Real] Pressure angle of the spur gear.

n_teeth

[int] Number of teeth of the spur gear.

tag

[str, default: None] User-defined label for identifying the face.

Returns**Sketch**

Revised sketch state ready for further sketch actions.

```
Sketch.tag(tag: str) → None
```

Add a tag to the active selection of sketch objects.

Parameters

tag`[str]` Tag to assign to the sketch objects.

`Sketch.plot(view_2d: bool = False, screenshot: str | None = None, use_frame: bool | None = None, selected_pd_objects: list[pyvista.PolyData] = None, **plotting_options: dict | None)`

Plot all objects of the sketch to the scene.

Parameters**view_2d**`[bool, default: False]` Whether to represent the plot in a 2D format.**screenshot**`[str, optional]` Path for saving a screenshot of the image that is being represented.**use_frame**`[bool, default: None]` Whether to enable the use of `frame`. The default is `None`, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.****plotting_options**`[dict, optional]` Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`Sketch.plot_selection(view_2d: bool = False, screenshot: str | None = None, use_frame: bool | None = None, **plotting_options: dict | None)`

Plot the current selection to the scene.

Parameters**view_2d**`[bool, default: False]` Whether to represent the plot in a 2D format.**screenshot**`[str, optional]` Path for saving a screenshot of the image that is being represented.**use_frame**`[bool, default: None]` Whether to enable the use of `frame`. The default is `None`, in which case the `ansys.tools.visualization_interface.USE_FRAME` global setting is used.****plotting_options**`[dict, optional]` Keyword arguments for plotting. For allowable keyword arguments, see the `Plotter.add_mesh` method.

`Sketch.sketch_polydata() → list[pyvista.PolyData]`

Get polydata configuration for all objects of the sketch.

Returns`list[PolyData]`

List of the polydata configuration for all edges and faces in the sketch.

`Sketch.sketch_polydata_faces() → list[pyvista.PolyData]`

Get polydata configuration for all faces of the sketch to the scene.

Returns`list[PolyData]`

List of the polydata configuration for faces in the sketch.

`Sketch.sketch_polydata_edges() → list[pyvista.PolyData]`

Get polydata configuration for all edges of the sketch to the scene.

Returns

`list[PolyData]`

List of the polydata configuration for edges in the sketch.

Description

Provides for creating and managing a sketch.

Module detail

`ansys.geometry.core.sketch.sketch.SketchObject`

Type to refer to both SketchEdge and SketchFace.

The slot.py module

Summary

Classes

<i>Slot</i> Provides for modeling a 2D slot.
--

Slot

```
class ansys.geometry.core.sketch.slot.Slot(center: ansys.geometry.core.math.point.Point2D, width: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, height: pint.Quantity | ansys.geometry.core.misc.measurements.Distance | ansys.geometry.core.typing.Real, angle: pint.Quantity | ansys.geometry.core.misc.measurements.Angle | ansys.geometry.core.typing.Real = 0)
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling a 2D slot.

Parameters

center: `:class: `Point2D` <ansys.geometry.core.math.point.Point2D>``
Center point of the slot.

width
`[Quantity | Distance | Real]` Width of the slot main body.

height
`[Quantity | Distance | Real]` Height of the slot.

angle
`[Quantity | Angle | Real, default: 0]` Placement angle for orientation alignment.

Overview

Properties

<i>center</i>	Center of the slot.
<i>width</i>	Width of the slot.
<i>height</i>	Height of the slot.
<i>perimeter</i>	Perimeter of the slot.
<i>area</i>	Area of the slot.
<i>visualization_polydata</i>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.slot import Slot
```

Property detail

property Slot.**center**: *ansys.geometry.core.math.point.Point2D*

Center of the slot.

property Slot.**width**: *pint.Quantity*

Width of the slot.

property Slot.**height**: *pint.Quantity*

Height of the slot.

property Slot.**perimeter**: *pint.Quantity*

Perimeter of the slot.

property Slot.**area**: *pint.Quantity*

Area of the slot.

property Slot.**visualization_polydata**: *pyvista.PolyData*

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

pyvista.PolyData

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a slot.

The trapezoid.py module

Summary

Classes

<i>Trapezoid</i>	Provides for modeling a 2D trapezoid.
------------------	---------------------------------------

Trapezoid

```
class ansys.geometry.core.sketch.trapezoid.Trapezoid(base_width: pint.Quantity | an-
                                                    sys.geometry.core.misc.measurements.Distance
                                                    | ansys.geometry.core.typing.Real, height:
                                                    pint.Quantity | an-
                                                    sys.geometry.core.misc.measurements.Distance
                                                    | ansys.geometry.core.typing.Real, base_angle:
                                                    pint.Quantity |
                                                    ansys.geometry.core.misc.measurements.Angle
                                                    | ansys.geometry.core.typing.Real,
                                                    base_asymmetric_angle: pint.Quantity |
                                                    ansys.geometry.core.misc.measurements.Angle
                                                    | ansys.geometry.core.typing.Real | None =
                                                    None, center:
                                                    ansys.geometry.core.math.point.Point2D =
                                                    ZERO_POINT2D, angle: pint.Quantity |
                                                    ansys.geometry.core.misc.measurements.Angle
                                                    | ansys.geometry.core.typing.Real = 0)
```

Bases: *ansys.geometry.core.sketch.face.SketchFace*

Provides for modeling a 2D trapezoid.

Parameters

base_width

[Quantity | Distance | Real] Width of the lower base of the trapezoid.

height

[Quantity | Distance | Real] Height of the slot.

base_angle

[Quantity | Distance | Real] Angle for trapezoid generation. Represents the angle on the base of the trapezoid.

base_asymmetric_angle

[Quantity | Angle | Real | None, default: None] Asymmetrical angles on each side of the trapezoid. The default is None, in which case the trapezoid is symmetrical. If provided, the trapezoid is asymmetrical and the right corner angle at the base of the trapezoid is set to the provided value.

center: Point2D, default: ZERO_POINT2D

Center point of the trapezoid.

angle

[Quantity | Angle | Real, default: 0] Placement angle for orientation alignment.

Notes

If an asymmetric base angle is defined, the base angle is applied to the left-most angle, and the asymmetric base angle is applied to the right-most angle.

Overview

Properties

<code>center</code>	Center of the trapezoid.
<code>base_width</code>	Width of the trapezoid.
<code>height</code>	Height of the trapezoid.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.trapezoid import Trapezoid
```

Property detail

property Trapezoid.**center**: `ansys.geometry.core.math.point.Point2D`

Center of the trapezoid.

property Trapezoid.**base_width**: `pint.Quantity`

Width of the trapezoid.

property Trapezoid.**height**: `pint.Quantity`

Height of the trapezoid.

property Trapezoid.**visualization_polydata**: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a trapezoid.

The triangle.py module

Summary

Classes

<code>Triangle</code>	Provides for modeling 2D triangles.
-----------------------	-------------------------------------

Triangle

```
class ansys.geometry.core.sketch.triangle.Triangle(point1: ansys.geometry.core.math.point.Point2D,
                                                    point2: ansys.geometry.core.math.point.Point2D,
                                                    point3: ansys.geometry.core.math.point.Point2D)
```

Bases: `ansys.geometry.core.sketch.face.SketchFace`

Provides for modeling 2D triangles.

Parameters

point1: Point2D

Point that represents a triangle vertex.

point2: Point2D

Point that represents a triangle vertex.

point3: Point2D

Point that represents a triangle vertex.

Overview

Properties

<code>point1</code>	Triangle vertex 1.
<code>point2</code>	Triangle vertex 2.
<code>point3</code>	Triangle vertex 3.
<code>visualization_polydata</code>	VTK polydata representation for PyVista visualization.

Import detail

```
from ansys.geometry.core.sketch.triangle import Triangle
```

Property detail

property Triangle.`point1`: `ansys.geometry.core.math.point.Point2D`

Triangle vertex 1.

property Triangle.`point2`: `ansys.geometry.core.math.point.Point2D`

Triangle vertex 2.

property Triangle.`point3`: `ansys.geometry.core.math.point.Point2D`

Triangle vertex 3.

property Triangle.`visualization_polydata`: `pyvista.PolyData`

VTK polydata representation for PyVista visualization.

The representation lies in the X/Y plane within the standard global Cartesian coordinate system.

Returns

`pyvista.PolyData`

VTK pyvista.Polydata configuration.

Description

Provides for creating and managing a triangle.

Description

PyAnsys Geometry sketch subpackage.

The tools package

Summary

Submodules

<code>check_geometry</code>	Module for repair tool message.
<code>measurement_tools</code>	Provides tools for measurement.
<code>prepare_tools</code>	Provides tools for preparing geometry for use with simulation.
<code>problem_areas</code>	The problem area definition.
<code>repair_tool_message</code>	Module for repair tool message.
<code>repair_tools</code>	Provides tools for repairing bodies.
<code>unsupported</code>	Unsupported functions for the PyAnsys Geometry library.

The `check_geometry.py` module

Summary

Classes

<code>GeometryIssue</code>	Provides return message for the repair tool methods.
<code>InspectResult</code>	Provides the result of the inspect geometry operation.

GeometryIssue

class `ansys.geometry.core.tools.check_geometry.GeometryIssue`(*message_type: str, message_id: str, message: str, edges: list[str], faces: list[str]*)

Provides return message for the repair tool methods.

Overview

Properties

<code>message_type</code>	The type of the message (warning, error, info).
<code>message_id</code>	The identifier for the message.
<code>message</code>	The content of the message.
<code>edges</code>	The List of edges (if any) that are part of the issue.
<code>faces</code>	The List of faces (if any) that are part of the issue.

Import detail

```
from ansys.geometry.core.tools.check_geometry import GeometryIssue
```

Property detail

property `GeometryIssue.message_type: str`

The type of the message (warning, error, info).

property `GeometryIssue.message_id: str`

The identifier for the message.

property `GeometryIssue.message: str`

The content of the message.

property `GeometryIssue.edges: list[str]`

The List of edges (if any) that are part of the issue.

property `GeometryIssue.faces: list[str]`

The List of faces (if any) that are part of the issue.

InspectResult

class `ansys.geometry.core.tools.check_geometry.InspectResult`(*grpc_client: ansys.geometry.core.connection.client.GrpcClient*,
body: ansys.geometry.core.designer.body.Body,
issues: list[GeometryIssue])

Provides the result of the inspect geometry operation.

Overview

Methods

<i>repair</i>	Repair the problem area.
---------------	--------------------------

Properties

<i>body</i>	The body for which issues are found.
<i>issues</i>	The list of issues for the body.

Import detail

```
from ansys.geometry.core.tools.check_geometry import InspectResult
```

Property detail

property `InspectResult.body: ansys.geometry.core.designer.body.Body`

The body for which issues are found.

property `InspectResult.issues: list[GeometryIssue]`

The list of issues for the body.

Method detail

`InspectResult.repair()` → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Repair the problem area.

Returns

RepairToolMessage

Message containing created and/or modified bodies.

Description

Module for repair tool message.

The `measurement_tools.py` module

Summary

Classes

<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,
                                                                    _internal_use: bool = False)
```

Measurement tools for PyAnsys Geometry.

Parameters

grpc_client
[GrpcClient] gRPC client to use for the measurement tools.

_internal_use
[bool, optional] Internal flag to prevent direct instantiation by users. This parameter is for internal use only.

Raises**GeometryRuntimeError**

If the class is instantiated directly by users instead of through the modeler.

Notes

This class should not be instantiated directly. Use `modeler.measurement_tools` instead.

Overview**Methods**

<code>min_distance_between_objects</code> Find the gap between two bodies.
--

Import detail

```
from ansys.geometry.core.tools.measurement_tools import MeasurementTools
```

Method detail

`MeasurementTools.min_distance_between_objects(object1: ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge, object2: ansys.geometry.core.designer.body.Body | ansys.geometry.core.designer.face.Face | ansys.geometry.core.designer.edge.Edge) → Gap`

Find the gap between two bodies.

Parameters**object1**

[Union[Body, Face, Edge]] First object to measure the gap.

object2

[Union[Body, Face, Edge]] Second object to measure the gap.

Returns**Gap**

Gap between two bodies.

Warning

This method is only available starting on Ansys release 24R2. Also, using Face and Edge objects as inputs requires a minimum Ansys release of 25R2.

Description

Provides tools for measurement.

The prepare_tools.py module

Summary

Classes

<i>EnclosureOptions</i>	Provides options related to enclosure creation.
<i>PrepareTools</i>	Prepare tools for PyAnsys Geometry.

EnclosureOptions

class ansys.geometry.core.tools.prepare_tools.**EnclosureOptions**

Provides options related to enclosure creation.

Options allow control on how the enclosure is inserted in the design.

Parameters

create_shared_topology

[*bool*, default: *False*] Whether shared topology should be applied after enclosure creation.

subtract_bodies

[*bool*, default: *True*] Whether the specified bodies for enclosure creation should be subtracted from the enclosure.

frame

[*Frame*, default: *None*] Frame used to orient the enclosure.

cushion_proportion

[*Real*, default: 0.25] A percentage of the minimum enclosure size. Determines the initial distance between the enclosed objects and the closest point of the enclosure to the objects.

Overview

Attributes

```
create_shared_topology
subtract_bodies
frame
cushion_proportion
```

Import detail

```
from ansys.geometry.core.tools.prepare_tools import EnclosureOptions
```

Attribute detail

EnclosureOptions.create_shared_topology: *bool* = *False*

EnclosureOptions.subtract_bodies: *bool* = *True*

EnclosureOptions.frame: *ansys.geometry.core.math.frame.Frame* = *None*

EnclosureOptions.cushion_proportion: *ansys.geometry.core.typing.Real* = 0.25

PrepareTools

class ansys.geometry.core.tools.prepare_tools.**PrepareTools**(*grpc_client: ansys.geometry.core.connection.GrpcClient*, *_internal_use: bool = False*)

Prepare tools for PyAnsys Geometry.

Parameters

grpc_client

[GrpcClient] Active supporting geometry service instance for design modeling.

_internal_use

[bool, optional] Internal flag to prevent direct instantiation by users. This parameter is for internal use only.

Raises

GeometryRuntimeError

If the class is instantiated directly by users instead of through the modeler.

Notes

This class should not be instantiated directly. Use `modeler.prepare_tools` instead.

Overview

Methods

<code>extract_volume_from_faces</code>	Extract a volume from input faces.
<code>extract_volume_from_edge_loops</code>	Extract a volume from input edge loops.
<code>remove_rounds</code>	Remove rounds from geometry.
<code>share_topology</code>	Share topology between the chosen bodies.
<code>enhanced_share_topology</code>	Share topology between the chosen bodies.
<code>find_logos</code>	Detect logos in geometry.
<code>find_and_remove_logos</code>	Detect and remove logos in geometry.
<code>detect_helices</code>	Detect helices in the given bodies.
<code>create_box_enclosure</code>	Create box enclosure around the given bodies.
<code>create_cylinder_enclosure</code>	Create cylinder enclosure around the given bodies.
<code>create_sphere_enclosure</code>	Create sphere enclosure around the given bodies.
<code>detect_sweepable_bodies</code>	Check if bodies are sweepable.

Import detail

```
from ansys.geometry.core.tools.prepare_tools import PrepareTools
```

Method detail

PrepareTools.extract_volume_from_faces(*sealing_faces: list[ansys.geometry.core.designer.face.Face]*, *inside_faces: list[ansys.geometry.core.designer.face.Face]*) → *list[ansys.geometry.core.designer.body.Body]*

Extract a volume from input faces.

Creates a volume (typically a flow volume) from a list of faces that seal the volume and one or more faces that define the wetted surface (inside faces of the solid).

Parameters**sealing_faces**

[list[Face]] List of faces that seal the volume.

inside_faces

[list[Face]] List of faces that define the interior of the solid.

Returns

list[Body]

List of created bodies.

Warning

This method is only available starting on Ansys release 25R1.

```
PrepareTools.extract_volume_from_edge_loops(sealing_edges:
                                             list[ansys.geometry.core.designer.edge.Edge], inside_faces:
                                             list[ansys.geometry.core.designer.face.Face] = None) →
                                             list[ansys.geometry.core.designer.body.Body]
```

Extract a volume from input edge loops.

Creates a volume (typically a flow volume) from a list of edge loops that seal the volume. and one or more faces that define the wetted surface (inside faces of the solid).

Parameters**sealing_edges**

[list[Edge]] List of faces that seal the volume.

inside_faces

[list[Face], optional] List of faces that define the interior of the solid (not always necessary).

Returns

list[Body]

List of created bodies.

Warning

This method is only available starting on Ansys release 25R1.

```
PrepareTools.remove_rounds(faces: list[ansys.geometry.core.designer.face.Face], auto_shrink: bool = False)
                           → bool
```

Remove rounds from geometry.

Tries to remove rounds from geometry. Faces to be removed are input to the method.

Parameters**round_faces**

[list[Face]] List of rounds faces to be removed

auto_shrink

[bool, default: False] Whether to shrink the geometry after removing rounds. Fills in the gaps left by the removed rounds.

Returns

bool

True if successful, False if failed.

PrepareTools.**share_topology**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *tol: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real = 0.0*, *preserve_instances: bool = False*) → **bool**

Share topology between the chosen bodies.

Parameters

bodies

[**list**[Body]] List of bodies to share topology between.

tol

[Distance | Quantity | Real] Maximum distance between bodies.

preserve_instances

[**bool**] Whether instances are preserved.

Returns

bool

True if successful, False if failed.

Warning

This method is only available starting on Ansys release 24R2.

PrepareTools.**enhanced_share_topology**(*bodies: list[ansys.geometry.core.designer.body.Body]*, *tol: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real = 0.0*, *preserve_instances: bool = False*) → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Share topology between the chosen bodies.

Parameters

bodies

[**list**[Body]] List of bodies to share topology between.

tol

[Distance | Quantity | Real] Maximum distance between bodies.

preserve_instances

[**bool**] Whether instances are preserved.

Returns

RepairToolMessage

Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

```
PrepareTools.find_logos(bodies: list[ansys.geometry.core.designer.body.Body] = None, min_height:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = None, max_height:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = None) →
    ansys.geometry.core.tools.problem_areas.LogoProblemArea
```

Detect logos in geometry.

Detects logos, using a list of bodies if provided. The logos are returned as a list of faces.

Parameters

bodies

[list[Body], optional] List of bodies where logos should be detected

min_height

[Distance | Quantity | Real, optional] The minimum height when searching for logos

max_height: Distance | Quantity | Real, optional

The minimum height when searching for logos

Returns

LogoProblemArea

Problem area with logo faces.

Warning

This method is only available starting on Ansys release 25R2.

```
PrepareTools.find_and_remove_logos(bodies: list[ansys.geometry.core.designer.body.Body] = None,
    min_height: ansys.geometry.core.misc.measurements.Distance |
    pint.Quantity | ansys.geometry.core.typing.Real = None, max_height:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = None) → bool
```

Detect and remove logos in geometry.

Detects and remove logos, using a list of bodies if provided.

Parameters

bodies

[list[Body], optional] List of bodies where logos should be detected and removed.

min_height

[Distance | Quantity | Real, optional] The minimum height when searching for logos

max_height: Distance | Quantity | Real, optional

The maximum height when searching for logos

Returns

Boolean value indicating whether the operation was successful.

Warning

This method is only available starting on Ansys release 25R2.

```

PrepareTools.detect_helices(bodies: list[ansys.geometry.core.designer.body.Body], min_radius:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = 0.0, max_radius:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = 100.0, fit_radius_error:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = 0.01) →
    dict[ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve,
    list[ansys.geometry.core.designer.edge.Edge]]

```

Detect helixes in the given bodies.

Parameters

bodies

[list[Body]] List of bodies to detect helixes in.

min_radius

[Distance, Quantity, or Real, default: 0.0] Minimum radius of the helix to be detected.

max_radius

[Distance, Quantity, or Real, default: 1e6] Maximum radius of the helix to be detected.

fit_radius_error

[Distance, Quantity, or Real, default: 0.01] Maximum fit radius error of the helix to be detected.

Returns

dict

Dictionary with key “helixes” containing a list of detected helixes. Each helix is represented as a dictionary with keys “trimmed_curve” and “edges”.

Warning

This method is only available starting on Ansys release 26R1.

```

PrepareTools.create_box_enclosure(bodies: list[ansys.geometry.core.designer.body.Body], x_low:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, x_high:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, y_low:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, y_high:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, z_low:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, z_high:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real, enclosure_options: EnclosureOptions)
    → list[ansys.geometry.core.designer.body.Body]

```

Create box enclosure around the given bodies.

Parameters

bodies

[list[Body]] List of bodies to create enclosure around.

x_low
[Distance | Quantity | Real] The lowest distance from the bodies in the x direction.

x_high
[Distance | Quantity | Real] The highest distance from the bodies in the x direction.

y_low
[Distance | Quantity | Real] The lowest distance from the bodies in the y direction.

y_high
[Distance | Quantity | Real] The highest distance from the bodies in the y direction.

z_low
[Distance | Quantity | Real] The lowest distance from the bodies in the z direction.

z_high
[Distance | Quantity | Real] The highest distance from the bodies in the z direction.

enclosure_options
[EnclosureOptions] Options that define how the enclosure is included in the design.

Returns

list[Body]
List of created bodies.

Warning

This method is only available starting on Ansys release 26R1.

```
PrepareTools.create_cylinder_enclosure(bodies: list[ansys.geometry.core.designer.body.Body],
                                       axial_distance_low:
                                       ansys.geometry.core.misc.measurements.Distance | pint.Quantity
                                       | ansys.geometry.core.typing.Real, axial_distance_high:
                                       ansys.geometry.core.misc.measurements.Distance | pint.Quantity
                                       | ansys.geometry.core.typing.Real, radial_distance:
                                       ansys.geometry.core.misc.measurements.Distance | pint.Quantity
                                       | ansys.geometry.core.typing.Real, enclosure_options:
                                       EnclosureOptions) →
                                       list[ansys.geometry.core.designer.body.Body]
```

Create cylinder enclosure around the given bodies.

Parameters

bodies
[list[Body]] List of bodies to create enclosure around.

axial_distance_low
[Distance | Quantity | Real] The lowest axial distance from the bodies.

axial_distance_high
[Distance | Quantity | Real] The highest axial distance from the bodies.

radial_distance
[Distance | Quantity | Real] The radial distance from the bodies.

enclosure_options
[EnclosureOptions] Options that define how the enclosure is included in the design.

Returns

list[Body]

List of created bodies.

Warning

This method is only available starting on Ansys release 26R1.

```
PrepareTools.create_sphere_enclosure(bodies: list[ansys.geometry.core.designer.body.Body],  
                                     radial_distance: ansys.geometry.core.misc.measurements.Distance |  
                                     pint.Quantity | ansys.geometry.core.typing.Real, enclosure_options:  
                                     EnclosureOptions) → list[ansys.geometry.core.designer.body.Body]
```

Create sphere enclosure around the given bodies.

Parameters**bodies**

[list[Body]] List of bodies to create enclosure around.

radial_distance

[Distance | Quantity | Real] The radial distance from the bodies.

enclosure_options

[EnclosureOptions] Options that define how the enclosure is included in the design.

Returns

list[Body]

List of created bodies.

Warning

This method is only available starting on Ansys release 26R1.

```
PrepareTools.detect_sweepable_bodies(bodies: list[ansys.geometry.core.designer.body.Body],  
                                     get_source_target_faces: bool = False) → list[tuple[bool,  
                                     list[ansys.geometry.core.designer.face.Face]]]
```

Check if bodies are sweepable.

Parameters**bodies**

[list[Body]] List of bodies to check.

get_source_target_faces

[bool] Whether to get source and target faces. By default, False.

Returns

list[tuple[bool, list[Face]]]

List of tuples, each containing a boolean indicating if the body is sweepable and a list of source and target faces if requested.

Description

Provides tools for preparing geometry for use with simulation.

The `problem_areas.py` module

Summary

Classes

<i>ProblemArea</i>	Represents problem areas.
<i>DuplicateFaceProblemAreas</i>	Provides duplicate face problem area definition.
<i>MissingFaceProblemAreas</i>	Provides missing face problem area definition.
<i>InexactEdgeProblemAreas</i>	Represents an inexact edge problem area with unique identifier and associated edges.
<i>ExtraEdgeProblemAreas</i>	Represents a extra edge problem area with unique identifier and associated edges.
<i>ShortEdgeProblemAreas</i>	Represents a short edge problem area with a unique identifier and associated edges.
<i>SmallFaceProblemAreas</i>	Represents a small face problem area with a unique identifier and associated faces.
<i>SplitEdgeProblemAreas</i>	Represents a split edge problem area with unique identifier and associated edges.
<i>StitchFaceProblemAreas</i>	Represents a stitch face problem area with unique identifier and associated faces.
<i>UnsimplifiedFaceProblemArea</i>	Represents a unsimplified face problem area with unique identifier and associated faces.
<i>InterferenceProblemAreas</i>	Represents an interference problem area with a unique identifier and associated bodies.
<i>LogoProblemArea</i>	Represents a logo problem area defined by a list of faces.

ProblemArea

class ansys.geometry.core.tools.problem_areas.**ProblemArea**(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient*)

Represents problem areas.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

Overview

Abstract methods

<i>fix</i> Fix problem area.

Methods

<code>build_repair_tool_message</code>	Build a repair tool message from the service response.
--	--

Properties

<code>id</code>	The id of the problem area.
-----------------	-----------------------------

Import detail

```
from ansys.geometry.core.tools.problem_areas import ProblemArea
```

Property detail

property `ProblemArea.id: str`

The id of the problem area.

Method detail

abstractmethod `ProblemArea.fix()`

Fix problem area.

`ProblemArea.build_repair_tool_message(response: dict) → an-`
`sys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Build a repair tool message from the service response.

Parameters

response

[dict] The response from the service containing information about the repair operation.

Returns

RepairToolMessage

A message containing the success status, created bodies, modified bodies, number of found problem areas, and number of repaired problem areas.

DuplicateFaceProblemAreas

class `ansys.geometry.core.tools.problem_areas.DuplicateFaceProblemAreas(id: str, grpc_client: an-`
`sys.geometry.core.connection.GrpcClient,`
`faces: list[ansys.geometry.core.designer.face.Face])`

Bases: `ProblemArea`

Provides duplicate face problem area definition.

Represents a duplicate face problem area with unique identifier and associated faces.

Parameters

id

[str] Server-defined ID for the problem area.

grpc_client

[GrpcClient] Active supporting geometry service instance for design modeling.

faces

[list[Face]] List of faces associated with the design.

Overview

Methods

fix Fix the problem area.

Properties

faces The list of faces connected to this problem area.

Import detail

```
from ansys.geometry.core.tools.problem_areas import DuplicateFaceProblemAreas
```

Property detail

property DuplicateFaceProblemAreas.faces: list[ansys.geometry.core.designer.face.Face]

The list of faces connected to this problem area.

Method detail

DuplicateFaceProblemAreas.fix() → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage

Fix the problem area.

Returns

message: RepairToolMessage

Message containing created and/or modified bodies.

MissingFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: list[ansys.geometry.core.designer.edge.Edge]*)

Bases: ProblemArea

Provides missing face problem area definition.

Parameters

id

[str] Server-defined ID for the problem area.

grpc_client

[GrpcClient] Active supporting geometry service instance for design modeling.

edges

[list[Edge]] List of edges associated with the design.

Overview

Methods

<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 MissingFaceProblemAreas
```

Property detail

property MissingFaceProblemAreas.**edges**: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

MissingFaceProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage

Message containing created and/or modified bodies.

InexactEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**InexactEdgeProblemAreas**(*id*: *str*, *grpc_client*: *ansys.geometry.core.connection.GrpcClient*, *edges*: *list[ansys.geometry.core.designer.edge.Edge]*)

Bases: ProblemArea

Represents an inexact edge problem area with unique identifier and associated edges.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

edges

[*list*[*Edge*]] List of edges associated with the design.

Overview

Methods

<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 InexactEdgeProblemAreas
```

Property detail

property InexactEdgeProblemAreas.**edges**: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

InexactEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

ExtraEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**ExtraEdgeProblemAreas**(*id*: *str*, *grpc_client*: *ansys.geometry.core.connection.GrpcClient*, *edges*: *list[ansys.geometry.core.designer.edge.Edge]*)

Bases: `ProblemArea`

Represents a extra edge problem area with unique identifier and associated edges.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

edges

[*list*[*Edge*]] List of edges associated with the design.

Overview

Methods

<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 ExtraEdgeProblemAreas
```

Property detail

property ExtraEdgeProblemAreas.**edges**: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

ExtraEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: RepairToolMessage

Message containing created and/or modified bodies.

ShortEdgeProblemAreas

class ansys.geometry.core.tools.problem_areas.**ShortEdgeProblemAreas**(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, edges: list[ansys.geometry.core.designer.edge.Edge]*)

Bases: ProblemArea

Represents a short edge problem area with a unique identifier and associated edges.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

edges

[*list*[*Edge*]] List of edges associated with the design.

Overview

Methods

<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 ShortEdgeProblemAreas
```

Property detail

property ShortEdgeProblemAreas.**edges**: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

ShortEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

SmallFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.**SmallFaceProblemAreas**(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, faces: list[ansys.geometry.core.designer.face.Face]*)

Bases: `ProblemArea`

Represents a small face problem area with a unique identifier and associated faces.

Parameters

id

[`str`] Server-defined ID for the problem area.

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

faces

[`list[Face]`] List of edges associated with the design.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>faces</i> The list of faces connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import SmallFaceProblemAreas
```

Property detail

property `SmallFaceProblemAreas.faces`: `list[ansys.geometry.core.designer.face.Face]`

The list of faces connected to this problem area.

Method detail

`SmallFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

SplitEdgeProblemAreas

class `ansys.geometry.core.tools.problem_areas.SplitEdgeProblemAreas`(*id*: *str*, *grpc_client*: *ansys.geometry.core.connection.GrpcClient*, *edges*: *list[ansys.geometry.core.designer.edge.Edge]*)

Bases: `ProblemArea`

Represents a split edge problem area with unique identifier and associated edges.

Parameters

id

[*str*] Server-defined ID for the problem area.

grpc_client

[*GrpcClient*] Active supporting geometry service instance for design modeling.

edges

[*list*[*Edge*]] List of edges associated with the design.

Overview

Methods

<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**: `list[ansys.geometry.core.designer.edge.Edge]`

The list of edges connected to this problem area.

Method detail

SplitEdgeProblemAreas.**fix**() → *ansys.geometry.core.tools.repair_tool_message.RepairToolMessage*

Fix the problem area.

Returns

message: `RepairToolMessage`

Message containing created and/or modified bodies.

StitchFaceProblemAreas

class ansys.geometry.core.tools.problem_areas.**StitchFaceProblemAreas**(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, bodies: list[ansys.geometry.core.designer.body.Body]*)

Bases: `ProblemArea`

Represents a stitch face problem area with unique identifier and associated faces.

Parameters

id

[`str`] Server-defined ID for the problem area.

grpc_client

[`GrpcClient`] Active supporting geometry service instance for design modeling.

bodies

[`list[Body]`] List of bodies associated with the design.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>bodies</i> The list of bodies connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import StitchFaceProblemAreas
```

Property detail

property `StitchFaceProblemAreas.bodies`: `list[ansys.geometry.core.designer.body.Body]`
The list of bodies connected to this problem area.

Method detail

`StitchFaceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`
Fix the problem area.

Returns

message: `RepairToolMessage`
Message containing created and/or modified bodies.

UnsimplifiedFaceProblemAreas

class `ansys.geometry.core.tools.problem_areas.UnsimplifiedFaceProblemAreas`(*id*: *str*,
grpc_client: *ansys.geometry.core.connection.GrpcClient*,
faces:
list[ansys.geometry.core.designer.face.Face])

Bases: `ProblemArea`

Represents a unsimplified face problem area with unique identifier and associated faces.

Parameters

id
[*str*] Server-defined ID for the problem area.

grpc_client
[*GrpcClient*] Active supporting geometry service instance for design modeling.

faces
[*list[Face]*] List of faces associated with the design.

Overview

Methods

<i>fix</i>	Fix the problem area.
------------	-----------------------

Properties

<i>faces</i>	The list of faces connected to this problem area.
--------------	---

Import detail

```
from ansys.geometry.core.tools.problem_areas import UnsimplifiedFaceProblemAreas
```

Property detail

property UnsimplifiedFaceProblemAreas.faces: list[ansys.geometry.core.designer.face.Face]
The list of faces connected to this problem area.

Method detail

UnsimplifiedFaceProblemAreas.fix() → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage
Fix the problem area.

Returns

message: RepairToolMessage
Message containing created and/or modified bodies.

InterferenceProblemAreas

class ansys.geometry.core.tools.problem_areas.InterferenceProblemAreas(*id: str, grpc_client: ansys.geometry.core.connection.GrpcClient, bodies: list[ansys.geometry.core.designer.body.Body]*)

Bases: ProblemArea

Represents an interference problem area with a unique identifier and associated bodies.

Parameters

id
[str] Server-defined ID for the problem area.

grpc_client
[GrpcClient] Active supporting geometry service instance for design modeling.

bodies
[list[Body]] List of bodies in the problem area.

Overview

Methods

<i>fix</i> Fix the problem area.

Properties

<i>bodies</i> The list of the bodies connected to this problem area.
--

Import detail

```
from ansys.geometry.core.tools.problem_areas import InterferenceProblemAreas
```

Property detail

property `InterferenceProblemAreas.bodies`: `list[ansys.geometry.core.designer.body.Body]`
The list of the bodies connected to this problem area.

Method detail

`InterferenceProblemAreas.fix()` → `ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`
Fix the problem area.

Returns

message: `RepairToolMessage`
Message containing created and/or modified bodies.

Notes

The current implementation does not properly track changes. The list of created and modified bodies are empty.

LogoProblemArea

```
class ansys.geometry.core.tools.problem_areas.LogoProblemArea(id: str, grpc_client: an-  
sys.geometry.core.connection.GrpcClient,  
face_ids: list[str])
```

Bases: `ProblemArea`

Represents a logo problem area defined by a list of faces.

Parameters

id
[str] Server-defined ID for the problem area.

grpc_client
[GrpcClient] Active supporting geometry service instance for design modeling.

faces
[list[str]] List of faces defining the logo problem area.

Overview

Methods

<i>fix</i>	Fix the problem area by deleting the logos.
------------	---

Properties

<i>face_ids</i>	The ids of the faces defining the logos.
-----------------	--

Import detail

```
from ansys.geometry.core.tools.problem_areas import LogoProblemArea
```

Property detail

property LogoProblemArea.**face_ids**: list[str]

The ids of the faces defining the logos.

Method detail

LogoProblemArea.**fix**() → bool

Fix the problem area by deleting the logos.

Returns

message: bool

Message that return whether the operation was successful.

Description

The problem area definition.

The repair_tool_message.py module

Summary

Classes

<i>RepairToolMessage</i>	Provides return message for the repair tool methods.
--------------------------	--

Functions

<i>create_repair_message_from_response</i>	Create a RepairToolMessage from a serialized response.
--	--

RepairToolMessage

```
class ansys.geometry.core.tools.repair_tool_message.RepairToolMessage(success: bool,
                                                                    created_bodies: list[str],
                                                                    modified_bodies:
                                                                    list[str], deleted_bodies:
                                                                    list[str] = None,
                                                                    craeted_components:
                                                                    list[str] = None,
                                                                    modified_components:
                                                                    list[str] = None,
                                                                    deleted_components:
                                                                    list[str] = None, found:
                                                                    int = -1, repaired: int =
                                                                    -1)
```

Provides return message for the repair tool methods.

Overview

Properties

<code>success</code>	The success of the repair operation.
<code>created_bodies</code>	The list of the created bodies after the repair operation.
<code>modified_bodies</code>	The list of the modified bodies after the repair operation.
<code>deleted_bodies</code>	The list of the deleted bodies after the repair operation.
<code>found</code>	Number of problem areas found for the repair operation.
<code>repaired</code>	Number of problem areas repaired during the repair operation.

Import detail

```
from ansys.geometry.core.tools.repair_tool_message import RepairToolMessage
```

Property detail

property `RepairToolMessage.success: bool`

The success of the repair operation.

property `RepairToolMessage.created_bodies: list[str]`

The list of the created bodies after the repair operation.

property `RepairToolMessage.modified_bodies: list[str]`

The list of the modified bodies after the repair operation.

property `RepairToolMessage.deleted_bodies: list[str]`

The list of the deleted bodies after the repair operation.

property `RepairToolMessage.found: int`

Number of problem areas found for the repair operation.

property `RepairToolMessage.repaired: int`

Number of problem areas repaired during the repair operation.

Description

Module for repair tool message.

Module detail

`ansys.geometry.core.tools.repair_tool_message.create_repair_message_from_response(response)`
→
RepairToolMessage

Create a RepairToolMessage from a serialized response.

Parameters

response

[serialized response] The serialized response object containing repair tool information.

Returns

RepairToolMessage

An instance of RepairToolMessage populated with data from the response.

The repair_tools.py module

Summary

Classes

<i>RepairTools</i>	Repair tools for PyAnsys Geometry.
--------------------	------------------------------------

RepairTools

class `ansys.geometry.core.tools.repair_tools.RepairTools`(*grpc_client*: *ansys.geometry.core.connection.GrpcClient*,
modeler:
ansys.geometry.core.modeler.Modeler,
_internal_use: *bool* = *False*)

Repair tools for PyAnsys Geometry.

Parameters

grpc_client

[GrpcClient] Active supporting geometry service instance for design modeling.

modeler

[Modeler] The parent modeler instance.

_internal_use

[bool, optional] Internal flag to prevent direct instantiation by users. This parameter is for internal use only.

Raises

GeometryRuntimeError

If the class is instantiated directly by users instead of through the modeler.

Notes

This class should not be instantiated directly. Use `modeler.repair_tools` instead.

Overview

Methods

<code>find_split_edges</code>	Find split edges in the given list of bodies.
<code>find_extra_edges</code>	Find the extra edges in the given list of bodies.
<code>find_inexact_edges</code>	Find inexact edges in the given list of bodies.
<code>find_short_edges</code>	Find the short edge problem areas.
<code>find_duplicate_faces</code>	Find the duplicate face problem areas.
<code>find_missing_faces</code>	Find the missing faces.
<code>find_small_faces</code>	Find the small face problem areas.
<code>find_stitch_faces</code>	Return the list of stitch face problem areas.
<code>find_simplify</code>	Detect faces in a body that can be simplified.
<code>find_interferences</code>	Find the interference problem areas.
<code>find_and_fix_short_edges</code>	Find and fix the short edge problem areas.
<code>find_and_fix_extra_edges</code>	Find and fix the extra edge problem areas.
<code>find_and_fix_split_edges</code>	Find and fix the split edge problem areas.
<code>find_and_fix_simplify</code>	Find and simplify the provided geometry.
<code>find_and_fix_stitch_faces</code>	Find and fix the stitch face problem areas.
<code>inspect_geometry</code>	Return a list of geometry issues organized by body.
<code>repair_geometry</code>	Attempt to repair the geometry for the given bodies.

Import detail

```
from ansys.geometry.core.tools.repair_tools import RepairTools
```

Method detail

`RepairTools.find_split_edges(bodies: list[ansys.geometry.core.designer.body.Body], angle: ansys.geometry.core.misc.measurements.Angle | pint.Quantity | ansys.geometry.core.typing.Real = None, length: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real = None) → 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

[Angle | Quantity | Real] The maximum angle between edges. By default, None.

length

[Distance | Quantity | Real] The maximum length of the edges. By default, None.

Returns

list[SplitEdgeProblemAreas]

List of objects representing split edge problem areas.

`RepairTools.find_extra_edges(bodies: list[ansys.geometry.core.designer.body.Body]) → list[ansys.geometry.core.tools.problem_areas.ExtraEdgeProblemAreas]`

Find the extra edges in the given list of bodies.

This method find the extra edge problem areas and returns a list of extra edge problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that extra edges are investigated on.

Returns

list[ExtraEdgeProblemArea]

List of objects representing extra edge problem areas.

`RepairTools.find_inexact_edges(bodies: list[ansys.geometry.core.designer.body.Body]) → list[ansys.geometry.core.tools.problem_areas.InexactEdgeProblemAreas]`

Find inexact edges in the given list of bodies.

This method find the inexact edge problem areas and returns a list of inexact edge problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that inexact edges are investigated on.

Returns

list[InExactEdgeProblemArea]

List of objects representing inexact edge problem areas.

`RepairTools.find_short_edges(bodies: list[ansys.geometry.core.designer.body.Body], length: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real = 0.0) → list[ansys.geometry.core.tools.problem_areas.ShortEdgeProblemAreas]`

Find the short edge problem areas.

This method finds the short edge problem areas and returns a list of these objects.

Parameters

bodies

[list[Body]] List of bodies that short edges are investigated on.

length

[Distance | Quantity | Real, optional] The maximum length of the edges. By default, 0.0.

Returns

list[ShortEdgeProblemAreas]

List of objects representing short edge problem areas.

`RepairTools.find_duplicate_faces(bodies: list[ansys.geometry.core.designer.body.Body]) → list[ansys.geometry.core.tools.problem_areas.DuplicateFaceProblemAreas]`

Find the duplicate face problem areas.

This method finds the duplicate face problem areas and returns a list of duplicate face problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that duplicate faces are investigated on.

Returns

list[DuplicateFaceProblemAreas]

List of objects representing duplicate face problem areas.

```
RepairTools.find_missing_faces(bodies: list[ansys.geometry.core.designer.body.Body], angle:
    ansys.geometry.core.misc.measurements.Angle | pint.Quantity |
    ansys.geometry.core.typing.Real | None = None, distance:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real | None = None) →
    list[ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas]
```

Find the missing faces.

This method find the missing face problem areas and returns a list of missing face problem areas objects.

Parameters**bodies**

[list[Body]] List of bodies that missing faces are investigated on.

angle

[Angle | Quantity | Real, optional] The minimum angle between faces. By default, None. This option is only used if the backend version is 26.1 or higher.

distance

[Distance | Quantity | Real, optional] The minimum distance between faces. By default, None. This option is only used if the backend version is 26.1 or higher.

Returns

list[MissingFaceProblemAreas]

List of objects representing missing face problem areas.

```
RepairTools.find_small_faces(bodies: list[ansys.geometry.core.designer.body.Body], area:
    ansys.geometry.core.misc.measurements.Area | pint.Quantity |
    ansys.geometry.core.typing.Real | None = None, width:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real | None = None) →
    list[ansys.geometry.core.tools.problem_areas.SmallFaceProblemAreas]
```

Find the small face problem areas.

This method finds and returns a list of ids of small face problem areas objects.

Parameters**bodies**

[list[Body]] List of bodies that small faces are investigated on.

area

[Area | Quantity | Real, optional] Maximum area of the faces. By default, None. This option is only used if the backend version is 26.1 or higher.

width

[Distance | Quantity | Real, optional] Maximum width of the faces. By default, None. This option is only used if the backend version is 26.1 or higher.

Returns

list[SmallFaceProblemAreas]

List of objects representing small face problem areas.

`RepairTools.find_stitch_faces(bodies: list[ansys.geometry.core.designer.body.Body], max_distance: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real | None = None) → list[ansys.geometry.core.tools.problem_areas.StitchFaceProblemAreas]`

Return the list of stitch face problem areas.

This method find the stitch face problem areas and returns a list of ids of stitch face problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that stitchable faces are investigated on.

max_distance

[Distance | Quantity | Real, optional] Maximum distance between faces. By default, None. This option is only used if the backend version is 26.1 or higher.

Returns

list[StitchFaceProblemAreas]

List of objects representing stitch face problem areas.

`RepairTools.find_simplify(bodies: list[ansys.geometry.core.designer.body.Body]) → list[ansys.geometry.core.tools.problem_areas.UnsimplifiedFaceProblemAreas]`

Detect faces in a body that can be simplified.

Parameters

bodies

[list[Body]] List of bodies to search.

Returns

list[UnsimplifiedFaceProblemAreas]

List of objects representing unsimplified face problem areas.

Warning

This method is only available starting on Ansys release 25R2.

`RepairTools.find_interferences(bodies: list[ansys.geometry.core.designer.body.Body], cut_smaller_body: bool = False) → list[ansys.geometry.core.tools.problem_areas.InterferenceProblemAreas]`

Find the interference problem areas.

This method finds and returns a list of ids of interference problem areas objects.

Parameters

bodies

[list[Body]] List of bodies that small faces are investigated on.

cut_smaller_body

[bool, optional] Whether to cut the smaller body if an intererence is found. By default, False.

Returns

`list[InterferenceProblemAreas]`

List of objects representing interference problem areas.

Warning

This method is only available starting on Ansys release 25R2.

`RepairTools.find_and_fix_short_edges(bodies: list[ansys.geometry.core.designer.body.Body], length: ansys.geometry.core.misc.measurements.Distance | pint.Quantity | ansys.geometry.core.typing.Real = 0.0, comprehensive_result: bool = False) → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Find and fix the short edge problem areas.

This method finds the short edges in the bodies and fixes them.

Parameters**bodies**

`[list[Body]]` List of bodies that short edges are investigated on.

length

`[Distance | Quantity | Real, optional]` The maximum length of the edges. By default, 0.0.

comprehensive_result

`[bool, optional]` Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

`RepairTools.find_and_fix_extra_edges(bodies: list[ansys.geometry.core.designer.body.Body], comprehensive_result: bool = False) → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Find and fix the extra edge problem areas.

This method finds the extra edges in the bodies and fixes them.

Parameters**bodies**

`[list[Body]]` List of bodies that short edges are investigated on.

comprehensive_result

`[bool, optional]` Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

```
RepairTools.find_and_fix_split_edges(bodies: list[ansys.geometry.core.designer.body.Body], angle:
    ansys.geometry.core.misc.measurements.Angle | pint.Quantity |
    ansys.geometry.core.typing.Real = 0.0, length:
    ansys.geometry.core.misc.measurements.Distance | pint.Quantity |
    ansys.geometry.core.typing.Real = 0.0, comprehensive_result: bool
    = False) →
    ansys.geometry.core.tools.repair_tool_message.RepairToolMessage
```

Find and fix the split edge problem areas.

This method finds the extra edges in the bodies and fixes them.

Parameters**bodies**

[list[Body]] List of bodies that split edges are investigated on.

angle

[Angle | Quantity | Real, optional] The maximum angle between edges. By default, 0.0.

length

[Distance | Quantity | Real, optional] The maximum length of the edges. By default, 0.0.

comprehensive_result

[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

```
RepairTools.find_and_fix_simplify(bodies: list[ansys.geometry.core.designer.body.Body],
    comprehensive_result: bool = False) →
    ansys.geometry.core.tools.repair_tool_message.RepairToolMessage
```

Find and simplify the provided geometry.

This method simplifies the provided geometry.

Parameters**bodies**

[list[Body]] List of bodies to be simplified.

comprehensive_result

[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

```
RepairTools.find_and_fix_stitch_faces(bodies: list[ansys.geometry.core.designer.body.Body],
                                     max_distance: ansys.geometry.core.misc.measurements.Distance |
                                     pint.Quantity | ansys.geometry.core.typing.Real | None = None,
                                     allow_multiple_bodies: bool = False, maintain_components: bool
                                     = True, check_for_coincidence: bool = False,
                                     comprehensive_result: bool = False) → an-
                                     sys.geometry.core.tools.repair_tool_message.RepairToolMessage
```

Find and fix the stitch face problem areas.

This method finds the stitchable faces and fixes them.

Parameters**bodies**

[list[Body]] List of bodies that stitchable faces are investigated on.

max_distance

[Distance | Quantity | Real, optional] The maximum distance between faces to be stitched. By default, 0.0001.

allow_multiple_bodies

[bool, optional] Whether to allow multiple bodies in the result. By default, False.

maintain_components

[bool, optional] Whether to stitch bodies within the components. By default, True.

check_for_coincidence

[bool, optional] Whether coincidence surfaces are searched. By default, False.

comprehensive_result

[bool, optional] Whether to fix all problem areas individually. By default, False.

Returns**RepairToolMessage**

Message containing number of problem areas found/fixed, created and/or modified bodies.

Warning

This method is only available starting on Ansys release 25R2.

```
RepairTools.inspect_geometry(bodies: list[ansys.geometry.core.designer.body.Body] = None) →
                             list[ansys.geometry.core.tools.check_geometry.InspectResult]
```

Return a list of geometry issues organized by body.

This method inspects the geometry and returns a list of the issues grouped by the body where they are found.

Parameters**bodies**

[list[Body]] List of bodies to inspect the geometry for. All bodies are inspected if the argument is not given.

Returns

`list[IssuesByBody]`

List of objects representing geometry issues and the bodies where issues are found.

Warning

This method is only available starting on Ansys release 25R2.

`RepairTools.repair_geometry(bodies: list[ansys.geometry.core.designer.body.Body] = None) → ansys.geometry.core.tools.repair_tool_message.RepairToolMessage`

Attempt to repair the geometry for the given bodies.

This method inspects the geometry for the given bodies and attempts to repair them.

Parameters

bodies

`[list[Body]]` List of bodies where to attempt to repair the geometry. All bodies are repaired if the argument is not given.

Returns

RepairToolMessage

Message containing success of the operation.

Warning

This method is only available starting on Ansys release 25R2.

Description

Provides tools for repairing bodies.

The unsupported.py module

Summary

Classes

<code>ExportIdData</code>	Data for exporting persistent ids.
<code>UnsupportedCommands</code>	Provides unsupported commands for PyAnsys Geometry.

Enums

<code>PersistentIdType</code>	Type of persistent id.
-------------------------------	------------------------

ExportIdData

`class ansys.geometry.core.tools.unsupported.ExportIdData`

Data for exporting persistent ids.

Overview

Attributes

<i>moniker</i>
<i>id_type</i>
<i>value</i>

Import detail

```
from ansys.geometry.core.tools.unsupported import ExportIdData
```

Attribute detail

`ExportIdData.moniker`: `str`

`ExportIdData.id_type`: `PersistentIdType`

`ExportIdData.value`: `str`

UnsupportedCommands

```
class ansys.geometry.core.tools.unsupported.UnsupportedCommands(grpc_client: an-  
    sys.geometry.core.connection.GrpcClient,  
    modeler: an-  
    sys.geometry.core.modeler.Modeler,  
    _internal_use: bool = False)
```

Provides unsupported commands for PyAnsys Geometry.

Parameters

`grpc_client`

[GrpcClient] gRPC client to use for the geometry commands.

`modeler`

[Modeler] Modeler instance to use for the geometry commands.

`_internal_use`

[bool, optional] Internal flag to prevent direct instantiation by users. This parameter is for internal use only.

Raises

`GeometryRuntimeError`

If the class is instantiated directly by users instead of through the modeler.

Notes

This class should not be instantiated directly. Use `modeler.unsupported` instead.

Overview

Methods

<code>set_export_id</code>	Set the persistent id for the moniker.
<code>set_multiple_export_ids</code>	Set multiple persistent ids for the monikers.
<code>get_body_occurrences_from_import_id</code>	Get all body occurrences whose master has the given import id.
<code>get_face_occurrences_from_import_id</code>	Get all face occurrences whose master has the given import id.
<code>get_edge_occurrences_from_import_id</code>	Get all edge occurrences whose master has the given import id.

Import detail

```
from ansys.geometry.core.tools.unsupported import UnsupportedCommands
```

Method detail

`UnsupportedCommands.set_export_id(moniker: str, id_type: PersistentIdType, value: str) → None`

Set the persistent id for the moniker.

Parameters

- moniker**
[str] Moniker to set the id for.
- id_type**
[PersistentIdType] Type of id.
- value**
[str] Id to set.

Warning

This method is only available starting on Ansys release 25R2.

`UnsupportedCommands.set_multiple_export_ids(export_data: list[ExportIdData]) → None`

Set multiple persistent ids for the monikers.

Parameters

- export_data**
[list[ExportIdData]] List of export data containing monikers, id types, and values.

Warning

This method is only available starting on Ansys release 26R1.

`UnsupportedCommands.get_body_occurrences_from_import_id(import_id: str, id_type: PersistentIdType)`

→

`list[ansys.geometry.core.designer.body.Body]`

Get all body occurrences whose master has the given import id.

Parameters

- import_id**
[str] Persistent id

id_type
[PersistentIdType] Type of id

Returns

list[Body]
List of body occurrences.

UnsupportedCommands.get_face_occurrences_from_import_id(import_id: str, id_type: PersistentIdType)
→
list[ansys.geometry.core.designer.face.Face]

Get all face occurrences whose master has the given import id.

Parameters

import_id
[str] Persistent id.
id_type
[PersistentIdType] Type of id.

Returns

list[Face]
List of face occurrences.

UnsupportedCommands.get_edge_occurrences_from_import_id(import_id: str, id_type: PersistentIdType)
→
list[ansys.geometry.core.designer.edge.Edge]

Get all edge occurrences whose master has the given import id.

Parameters

import_id
[str] Persistent id.
id_type
[PersistentIdType] Type of id.

Returns

list[Edge]
List of edge occurrences.

PersistentIdType

class ansys.geometry.core.tools.unsupported.PersistentIdType(*args, **kws)

Bases: `enum.Enum`

Type of persistent id.

Overview

Attributes

<i>PNAME</i>
<i>PRIME_ID</i>

Import detail

```
from ansys.geometry.core.tools.unsupported import PersistentIdType
```

Attribute detail

PersistentIdType.PNAME = 1

PersistentIdType.PRIME_ID = 700

Description

Unsupported functions for the PyAnsys Geometry library.

Description

PyAnsys Geometry tools subpackage.

The errors.py module

Summary

Exceptions

<i>GeometryRuntimeError</i>	Provides error message when Geometry service passes a runtime error.
<i>GeometryExitedError</i>	Provides error message to raise when Geometry service has exited.

Functions

<i>handler</i>	Pass signal to the custom interrupt handler.
<i>protect_grpc</i>	Capture gRPC exceptions and raise a more succinct error message.

Constants

SIGINT_TRACKER

GeometryRuntimeError

exception ansys.geometry.core.errors.GeometryRuntimeError

Bases: RuntimeError

Provides error message when Geometry service passes a runtime error.

Import detail

```
from ansys.geometry.core.errors import GeometryRuntimeError
```

GeometryExitedError

exception `ansys.geometry.core.errors.GeometryExitedError(msg='Geometry service has exited.')`

Bases: `RuntimeError`

Provides error message to raise when Geometry service has exited.

Parameters

msg

[`str`, default: “Geometry service has exited.”] Message to raise.

Import detail

```
from ansys.geometry.core.errors import GeometryExitedError
```

Description

Provides PyAnsys Geometry-specific errors.

Module detail

`ansys.geometry.core.errors.handler(sig, frame)`

Pass signal to the custom interrupt handler.

`ansys.geometry.core.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`.

`ansys.geometry.core.errors.SIGINT_TRACKER = []`

The `logger.py` module

Summary

Classes

<code>PyGeometryCustomAdapter</code>	Keeps the reference to the Geometry service instance name dynamic.
<code>PyGeometryPercentStyle</code>	Provides a common messaging style.
<code>PyGeometryFormatter</code>	Provides a <code>Formatter</code> class for overwriting default format styles.
<code>InstanceFilter</code>	Ensures that the <code>instance_name</code> record always exists.
<code>Logger</code>	Provides the logger used for each PyAnsys Geometry session.

Functions

<code>addfile_handler</code>	Add a file handler to the input.
<code>add_stdout_handler</code>	Add a stdout handler to the logger.

Attributes

<code>string_to_loglevel</code>

Constants

<code>LOG_LEVEL</code>
<code>FILE_NAME</code>
<code>DEBUG</code>
<code>INFO</code>
<code>WARN</code>
<code>ERROR</code>
<code>CRITICAL</code>
<code>STDOUT_MSG_FORMAT</code>
<code>FILE_MSG_FORMAT</code>
<code>DEFAULT_STDOUT_HEADER</code>
<code>DEFAULT_FILE_HEADER</code>
<code>NEW_SESSION_HEADER</code>
<code>LOG</code>

PyGeometryCustomAdapter

class ansys.geometry.core.logger.**PyGeometryCustomAdapter**(*logger, extra=None*)

Bases: `logging.LoggerAdapter`

Keeps the reference to the Geometry service instance name dynamic.

If you use the standard approach, which is supplying *extra* input to the logger, you must input Geometry service instances each time you do a log.

Using adapters, you only need to specify the Geometry service instance that you are referring to once.

Overview

Methods

<code>process</code>	Process the logging message and keyword arguments passed in to
<code>log_to_file</code>	Add a file handler to the logger.
<code>log_to_stdout</code>	Add a standard output handler to the logger.
<code>setLevel</code>	Change the log level of the object and the attached handlers.

Attributes

<code>level</code>
<code>file_handler</code>
<code>stdout_handler</code>
<code>logger</code>
<code>std_out_handler</code>

Import detail

```
from ansys.geometry.core.logger import PyGeometryCustomAdapter
```

Attribute detail

`PyGeometryCustomAdapter.level = None`

`PyGeometryCustomAdapter.file_handler = None`

`PyGeometryCustomAdapter.stdout_handler = None`

`PyGeometryCustomAdapter.logger`

`PyGeometryCustomAdapter.std_out_handler`

Method detail

`PyGeometryCustomAdapter.process(msg, kwargs)`

Process the logging message and keyword arguments passed in to a logging call to insert contextual information. You can either manipulate the message itself, the keyword args or both. Return the message and kwargs modified (or not) to suit your needs.

Normally, you'll only need to override this one method in a `LoggerAdapter` subclass for your specific needs.

`PyGeometryCustomAdapter.log_to_file(filename: str = FILE_NAME, level: int = LOG_LEVEL)`

Add a file handler to the logger.

Parameters

filename

[`str`, default: "pyansys-geometry.log"] Name of the file to write log messages to.

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`PyGeometryCustomAdapter.log_to_stdout(level=LOG_LEVEL)`

Add a standard output handler to the logger.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

`PyGeometryCustomAdapter.setLevel(level='DEBUG')`

Change the log level of the object and the attached handlers.

Parameters

level

[`int`, default: 10] Level of logging. The default is 10, in which case the `logging.DEBUG` level is used.

PyGeometryPercentStyle

class ansys.geometry.core.logger.**PyGeometryPercentStyle**(*fmt*, * (*Keyword-only parameters separator* (PEP 3102)), *defaults=None*)

Bases: `logging.PercentStyle`

Provides a common messaging style.

Import detail

```
from ansys.geometry.core.logger import PyGeometryPercentStyle
```

PyGeometryFormatter

class ansys.geometry.core.logger.**PyGeometryFormatter**(*fmt=STDOUT_MSG_FORMAT*,
datefmt=None, *style='%'*, *validate=True*,
defaults=None)

Bases: `logging.Formatter`

Provides a `Formatter` class for overwriting default format styles.

Import detail

```
from ansys.geometry.core.logger import PyGeometryFormatter
```

InstanceFilter

class ansys.geometry.core.logger.**InstanceFilter**(*name=""*)

Bases: `logging.Filter`

Ensures that the `instance_name` record always exists.

Overview

Methods

<i>filter</i>	Ensure that the <code>instance_name</code> attribute is always present.
---------------	---

Import detail

```
from ansys.geometry.core.logger import InstanceFilter
```

Method detail

`InstanceFilter.filter(record)`

Ensure that the `instance_name` attribute is always present.

Logger

class ansys.geometry.core.logger.**Logger**(*level=logging.DEBUG, to_file=False, to_stdout=True, filename=FILE_NAME*)

Provides the logger used for each PyAnsys Geometry session.

This class allows you to add handlers to the logger to output messages to a file or to the standard output (stdout).

Parameters

level

[**int**, default: 10] Logging level to filter the message severity allowed in the logger. The default is 10, in which case the logging.DEBUG level is used.

to_file

[**bool**, default: **False**] Whether to write log messages to a file.

to_stdout

[**bool**, default: **True**] Whether to write log messages to the standard output.

filename

[**str**, default: "pyansys-geometry.log"] Name of the file to write log log messages to.

Examples

Demonstrate logger usage from the `Modeler` instance, which is automatically created when a Geometry service instance is created.

```
>>> from ansys.geometry.core import Modeler
>>> modeler = Modeler(loglevel="DEBUG")
>>> modeler._log.info("This is a useful message")
INFO - - <ipython-input-24-80df150fe31f> - <module> - This is LOG debug message.
```

Import the global PyAnsys Geometry logger and add a file output handler.

```
>>> import os
>>> from ansys.geometry.core import LOG
>>> file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
>>> LOG.log_to_file(file_path)
```

Overview

Methods

<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>
<code>logger</code>
<code>level</code>
<code>debug</code>
<code>info</code>
<code>warning</code>
<code>error</code>
<code>critical</code>
<code>log</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 = None`

`Logger.std_out_handler = None`

`Logger.logger`

`Logger.level = 0`

`Logger.debug`

`Logger.info`

`Logger.warning`

`Logger.error`

`Logger.critical`

`Logger.log`

Method detail

`Logger.log_to_file(filename=FILE_NAME, level=LOG_LEVEL)`

Add a file handler to the logger.

Parameters

filename

[`str`, default: "pyansys-geometry.log"] Name of the file to write log messages to.

level

[[int](#), default: 10] Level of logging. The default is 10, in which case the logging.DEBUG level is used.

Examples

Write to the "pyansys-geometry.log" file in the current working directory:

```
>>> from ansys.geometry.core import LOG
>>> import os
>>> file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
>>> LOG.log_to_file(file_path)
```

`Logger.log_to_stdout(level=LOG_LEVEL)`

Add the standard output handler to the logger.

Parameters**level**

[[int](#), default: 10] Level of logging. The default is 10, in which case the logging.DEBUG level is used.

`Logger.setLevel(level='DEBUG')`

Change the log level of the object and the attached handlers.

`Logger.add_child_logger(suffix: str, level: str | None = None)`

Add a child logger to the main logger.

This logger is more general than an instance logger, which is designed to track the state of Geometry service instances.

If the logging level is in the arguments, a new logger with a reference to the `_global` logger handlers is created instead of a child logger.

Parameters**suffix**

[[str](#)] Name of the child logger.

level

[[str](#), default: `None`] Level of logging.

Returns

[logging.Logger](#)

Logger class.

`Logger.add_instance_logger(name: str, client_instance: ansys.geometry.core.connection.client.GrpcClient, level: int | None = None) → PyGeometryCustomAdapter`

Add a logger for a Geometry service instance.

The Geometry service instance logger is a logger with an adapter that adds contextual information such as the Geometry service instance name. This logger is returned, and you can use it to log events as a normal logger. It is stored in the `_instances` field.

Parameters**name**

[[str](#)] Name for the new instance logger.

client_instance

[GrpcClient] Geometry service GrpcClient object, which should contain the `get_name` method.

level

[int, default: `None`] Level of logging.

Returns**PyGeometryCustomAdapter**

Logger adapter customized to add Geometry service information to the logs. You can use this class to log events in the same way you would with the `Logger` class.

`Logger.__getitem__(key)`

Overload the access method by item for the `Logger` class.

`Logger.add_handling_uncaught_exceptions(logger)`

Redirect the output of an exception to a logger.

Parameters**logger**

[str] Name of the logger.

Description

Provides a general framework for logging in PyAnsys Geometry.

This module is built on the [Logging facility for Python](#). It is not intended to replace the standard Python logging library but rather provide a way to interact between its `logging` class and PyAnsys Geometry.

The loggers used in this module include the name of the instance, which is intended to be unique. This name is printed in all active outputs and is used to track the different Geometry service instances.

Logger usage**Global logger**

There is a global logger named `PyAnsys_Geometry_global` that is created when `ansys.geometry.core.__init__` is called. If you want to use this global logger, you must call it at the top of your module:

```
from ansys.geometry.core import LOG
```

You can rename this logger to avoid conflicts with other loggers (if any):

```
from ansys.geometry.core import LOG as logger
```

The default logging level of `LOG` is `ERROR`. You can change this level and output lower-level messages with this code:

```
LOG.logger.setLevel("DEBUG")
LOG.file_handler.setLevel("DEBUG") # If present.
LOG.stdout_handler.setLevel("DEBUG") # If present.
```

Alternatively, you can ensure that all the handlers are set to the input log level with this code:

```
LOG.setLevel("DEBUG")
```

This logger does not log to a file by default. If you want, you can add a file handler with this code:

```
import os

file_path = os.path.join(os.getcwd(), "pyansys-geometry.log")
LOG.log_to_file(file_path)
```

This also sets the logger to be redirected to this file. If you want to change the characteristics of this global logger from the beginning of the execution, you must edit the `__init__` file in the directory `ansys.geometry.core`.

To log using this logger, call the desired method as a normal logger with:

```
>>> import logging
>>> from ansys.geometry.core.logging import Logger
>>> LOG = Logger(level=logging.DEBUG, to_file=False, to_stdout=True)
>>> LOG.debug("This is LOG debug message.")

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

```
ansys.geometry.core.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.

`ansys.geometry.core.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.

`ansys.geometry.core.logger.LOG_LEVEL = 10`

`ansys.geometry.core.logger.FILE_NAME = 'pyansys-geometry.log'`

`ansys.geometry.core.logger.DEBUG = 10`

`ansys.geometry.core.logger.INFO = 20`

`ansys.geometry.core.logger.WARN = 30`

`ansys.geometry.core.logger.ERROR = 40`

`ansys.geometry.core.logger.CRITICAL = 50`

`ansys.geometry.core.logger.STDOUT_MSG_FORMAT = '%(levelname)s - %(instance_name)s - %(module)s - %(funcName)s - %(message)s'`

`ansys.geometry.core.logger.FILE_MSG_FORMAT = '%(levelname)s - %(instance_name)s - %(module)s - %(funcName)s - %(message)s'`

`ansys.geometry.core.logger.DEFAULT_STDOUT_HEADER = Multiline-String`

```
"""
LEVEL - INSTANCE NAME - MODULE - FUNCTION - MESSAGE
"""
```

`ansys.geometry.core.logger.DEFAULT_FILE_HEADER = Multiline-String`

```
"""
LEVEL - INSTANCE NAME - MODULE - FUNCTION - MESSAGE
"""
```

`ansys.geometry.core.logger.NEW_SESSION_HEADER`

`ansys.geometry.core.logger.LOG`

`ansys.geometry.core.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 = pygeom_defaults.DEFAULT_HOST, port: str | int
    = pygeom_defaults.DEFAULT_PORT, channel:
    grpc.Channel | None = None, remote_instance:
    ansys.platform.instancemanagement.Instance | None = None,
    docker_instance: an-
    sys.geometry.core.connection.docker_instance.LocalDockeInstance
    | None = None, product_instance: an-
    sys.geometry.core.connection.product_instance.ProductInstance
    | None = None, timeout: ansys.geometry.core.typing.Real =
    120, logging_level: int = logging.INFO, logging_file:
    pathlib.Path | str | None = None, proto_version: str | None =
    None, transport_mode: str | None = None, uds_id: str | None = None,
    certs_dir: pathlib.Path | str | None = None)
```

Provides for interacting with an open session of the Geometry service.

Parameters

host

[*str*, default: `DEFAULT_HOST`] Host where the server is running.

port

[*str* | *int*, default: `DEFAULT_PORT`] Port number where the server is running.

channel

[*Channel*, default: `None`] gRPC channel for server communication.

remote_instance

[*ansys.platform.instancemanagement.Instance*, default: `None`] Corresponding remote instance when the Geometry service is launched using `PyPIM`. This instance is deleted when the `GrpcClient.close` method is called.

docker_instance

[*LocalDockeInstance*, 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.

proto_version

[`str` | `None`, default: `None`] Protocol version to use for communication with the server. If `None`, `v0` is used. Available versions are “v0”, “v1”, etc.

transport_mode

[`str` | `None`] Transport mode selected. Needed if `channel` is not provided. Options are: “insecure”, “uds”, “wnua”, “mtls”.

uds_dir

[`Path` | `str` | `None`] Directory to use for Unix Domain Sockets (UDS) transport mode. By default `None` and thus it will use the “~/conn” folder.

uds_id

[`str` | `None`] Optional ID to use for the UDS socket filename. By default `None` and thus it will use “apostas_socket.sock”. Otherwise, the socket filename will be “apostas_socket-<uds_id>.sock”.

certs_dir

[`Path` | `str` | `None`] Directory to use for TLS certificates. By default `None` and thus search for the “ANSYS_GRPC_CERTIFICATES” environment variable. If not found, it will use the “certs” folder assuming it is in the current working directory.

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>	Access the client’s close method.
<code>exit</code>	Access the client’s close method.
<code>open_file</code>	Open a file.
<code>run_discovery_script_file</code>	Run a Discovery script file.
<code>get_service_logs</code>	Get the service logs.

Properties

<code>client</code>	Modeler instance client.
<code>design</code>	Retrieve the design within the modeler workspace.
<code>repair_tools</code>	Access to repair tools.
<code>prepare_tools</code>	Access to prepare tools.
<code>measurement_tools</code>	Access to measurement tools.
<code>geometry_commands</code>	Access to geometry commands.
<code>unsupported</code>	Access to unsupported commands.

Special methods

<code>__repr__</code>	Represent the modeler as a string.
-----------------------	------------------------------------

Import detail

```
from ansys.geometry.core.modeler import Modeler
```

Property detail

property `Modeler.client`: *ansys.geometry.core.connection.client.GrpcClient*

Modeler instance client.

property `Modeler.design`: *ansys.geometry.core.designer.design.Design*

Retrieve the design within the modeler workspace.

property `Modeler.repair_tools`: *ansys.geometry.core.tools.repair_tools.RepairTools*

Access to repair tools.

property `Modeler.prepare_tools`: *ansys.geometry.core.tools.prepare_tools.PrepareTools*

Access to prepare tools.

property `Modeler.measurement_tools`:
ansys.geometry.core.tools.measurement_tools.MeasurementTools

Access to measurement tools.

Notes

This property is only available starting on Ansys release 24R2.

property `Modeler.geometry_commands`:
ansys.geometry.core.designer.geometry_commands.GeometryCommands

Access to geometry commands.

property `Modeler.unsupported`: *ansys.geometry.core.tools.unsupported.UnsupportedCommands*

Access to unsupported commands.

Method detail

`Modeler.create_design(name: str) → ansys.geometry.core.designer.design.Design`

Initialize a new design with the connected client.

Parameters

name

[str] Name for the new design.

Returns

Design

Design object created on the server.

`Modeler.get_active_design(sync_with_backend: bool = True) → ansys.geometry.core.designer.design.Design`

Get the active design on the modeler object.

Parameters

sync_with_backend

[*bool*, default: *True*] Whether to sync the active design with the remote service. If set to *False*, the active design may be out-of-sync with the remote service. This is useful when the active design is known to be up-to-date.

Returns

Design

Design object already existing on the modeler.

Modeler.read_existing_design() → *ansys.geometry.core.designer.design.Design*

Read the existing design on the service with the connected client.

Returns

Design

Design object already existing on the server.

Modeler.close(close_design: *bool* = *True*) → *None*

Access the client's close method.

Parameters

close_design

[*bool*, default: *True*] Whether to close the design before closing the client.

Modeler.exit(close_design: *bool* = *True*) → *None*

Access the client's close method.

Parameters

close_design

[*bool*, default: *True*] Whether to close the design before closing the client.

Notes

This method is calling the same method as `close()`.

Modeler.open_file(file_path: *str* | *pathlib.Path*, upload_to_server: *bool* = *True*, import_options: *ansys.geometry.core.misc.options.ImportOptions* = *ImportOptions*(), import_options_definitions: *ansys.geometry.core.misc.options.ImportOptionsDefinitions* = *ImportOptionsDefinitions*()) → *ansys.geometry.core.designer.design.Design*

Open a file.

This method imports a design into the service. On Windows and Linux, `.scdocx`, `.dsco`, and reader formats are supported. Please see notes for supported reader formats.

If the file is a shattered assembly with external references, the whole containing folder will need to be uploaded. Ensure proper folder structure in order to prevent the uploading of unnecessary files.

Parameters

file_path

[*str*, *Path*] Path of the file to open. The extension of the file must be included.

upload_to_server

[*bool*] True if the service is running on a remote machine. If service is running on the local machine, set to *False*, as there is no reason to upload the file.

import_options

[*ImportOptions*] Import options that toggle certain features when opening a file.

Returns

Design

Newly imported design.

Notes

Format and latest supported version

- AutoCAD 2024
- CATIA V5 2024
- CATIA V6 2024
- Creo Parametric 11
- IGES 5.3
- Inventor 2025
- JT 10.10
- NX 2412
- Rhino 8
- Solid Edge 2025
- SOLIDWORKS 2025
- STEP AP242

`Modeler.__repr__()` → `str`

Represent the modeler as a string.

`Modeler.run_discovery_script_file(file_path: str | pathlib.Path, script_args: dict[str, str] | None = None, import_design: bool = False, api_version: int | str | ansys.geometry.core.connection.backend.ApiVersions = None) → tuple[dict[str, str], ansys.geometry.core.designer.design.Design | None]`

Run a Discovery script file.

Note

If arguments are passed to the script, they must be in the form of a dictionary. On the server side, the script will receive the arguments as a dictionary of strings, under the variable name `argsDict`. For example, if the script is called with the arguments `run_discovery_script_file(..., script_args = {"length": "20"}, ...)`, the script will receive the dictionary `argsDict` with the key-value pair `{"length": "20"}`.

Note

If an output is expected from the script, it will be returned as a dictionary of strings. The keys and values of the dictionary are the variables and their values that the script returns. However, it is necessary that the script creates a dictionary called `result` with the variables and their values that are expected to be returned. For example, if the script is expected to return the number of bodies in the design, the script should create a dictionary called `result` with the key-value pair `{"numBodies": numBodies}`, where `numBodies` is the number of bodies in the design.

The implied API version of the script should match the API version of the running Geometry Service. DMS API versions 24.1 and later are supported. DMS is a Windows-based modeling service that has been containerized to ease distribution, execution, and remotability operations.

Parameters

file_path

[`str` | `Path`] Path of the file. The extension of the file must be included.

script_args

[`dict`[`str`, `str`], optional.] Arguments to pass to the script. By default, `None`.

import_design

[`bool`, optional.] Whether to refresh the current design from the service. When the script is expected to modify the existing design, set this to `True` to retrieve up-to-date design data. When this is set to `False` (default) and the script modifies the current design, the design may be out-of-sync. By default, `False`.

api_version

[`int` | `str` | `ApiVersions`, optional] The scripting API version to use. For example, version 24.1 can be passed as an integer 241, a string “241” or using the `ansys.geometry.core.connection.backend.ApiVersions` enum class. By default, `None`. When specified, the service will attempt to run the script with the specified API version. If the API version is not supported, the service will raise an error. If you are using Discovery or SpaceClaim, the product will determine the API version to use, so there is no need to specify this parameter.

Returns

`dict`[`str`, `str`]

Values returned from the script.

Design, optional

Up-to-date current design. This is only returned if `import_design=True`.

Raises

GeometryRuntimeError

If the Discovery script fails to run. Otherwise, assume that the script ran successfully.

Notes

The Ansys Geometry Service only supports scripts that are of the same version as the running service. Any `api_version` input will be ignored.

`Modeler.get_service_logs(all_logs: bool = False, dump_to_file: bool = False, logs_folder: str | pathlib.Path | None = None) → str | dict[str, str] | pathlib.Path`

Get the service logs.

Parameters

all_logs

[`bool`, default: `False`] Flag indicating whether all logs should be retrieved. By default, only the current logs are retrieved.

dump_to_file

[`bool`, default: `False`] Flag indicating whether the logs should be dumped to a file. By default, the logs are not dumped to a file.

logs_folder

[[str](#), Path or [None](#), default: [None](#)] Name of the folder where the logs should be dumped. This parameter is only used if the `dump_to_file` parameter is set to `True`.

Returns

[str](#)

Service logs as a string. This is returned if the `dump_to_file` parameter is set to `False`.

[dict\[str, str\]](#)

Dictionary containing the logs. The keys are the logs names, and the values are the logs as strings. This is returned if the `all_logs` parameter is set to `True` and the `dump_to_file` parameter is set to `False`.

Path

Path to the folder containing the logs (if the `all_logs` parameter is set to `True`) or the path to the log file (if only the current logs are retrieved). The `dump_to_file` parameter must be set to `True`.

Notes

This property is only available starting on Ansys release 25R1.

Description

Provides for interacting with the Geometry service.

The `typing.py` module

Summary

Attributes

Real	Type used to refer to both integers and floats as possible values.
RealSequence	Type used to refer to Real types as a Sequence type.

Description

Provides typing of values for PyAnsys Geometry.

Module detail

`ansys.geometry.core.typing.Real`

Type used to refer to both integers and floats as possible values.

`ansys.geometry.core.typing.RealSequence`

Type used to refer to Real types as a Sequence type.

Notes

[numpy.ndarrays](#) are also accepted because they are the overlaying data structure behind most PyAnsys Geometry objects.

7.1.2 Description

PyAnsys Geometry is a Python wrapper for the Ansys Geometry service.

7.1.3 Module detail

`ansys.geometry.core.USE_SERVICE_COLORS: bool = False`

Global constant for checking whether to use service colors for plotting purposes. If set to False, the default colors will be used (speed gain).

`ansys.geometry.core.DISABLE_MULTIPLE_DESIGN_CHECK: bool = True`

Deprecated constant. Use `DISABLE_ACTIVE_DESIGN_CHECK` instead.

`ansys.geometry.core.DISABLE_ACTIVE_DESIGN_CHECK: bool = False`

Global constant for disabling the `ensure_design_is_active` check.

Only set this to false if you are sure you want to disable this check and your objects will always exist on the server side.

`ansys.geometry.core.DOCUMENTATION_BUILD: bool`

Global flag for the documentation to use the proper PyVista Jupyter backend.

`ansys.geometry.core.USE_TRACKER_TO_UPDATE_DESIGN: bool = False`

Global constant for checking whether to use the tracker to update designs.

`ansys.geometry.core.__version__`

PyAnsys Geometry version.

EXAMPLES

These examples demonstrate the behavior and usage of PyAnsys Geometry.

8.1 PyAnsys Geometry 101 examples

These examples demonstrate basic operations you can perform with PyAnsys Geometry.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.1.1 PyAnsys Geometry 101: Math

The `math` module is the foundation of PyAnsys Geometry. This module is built on top of `NumPy`, one of the most renowned mathematical Python libraries.

This example shows some of the main PyAnsys Geometry math objects and demonstrates why they are important prior to doing more exciting things in PyAnsys Geometry.

Perform required imports

Perform the required imports.

```
[1]: import numpy as np

from ansys.geometry.core.math import Plane, Point2D, Point3D, Vector2D, Vector3D, \
    UnitVector3D
```

Create points and vectors

Everything starts with `Point` and `Vector` objects, which can each be defined in a 2D or 3D form. These objects inherit from NumPy's `ndarray`, providing them with enhanced functionalities. When creating these objects, you must remember to pass in the arguments as a list (that is, with brackets `[]`).

Create 2D and 3D point and vectors.

```
Point3D([x, y, z])
Point2D([x, y])
```

(continues on next page)

(continued from previous page)

```
Vector3D([x, y, z])
Vector2D([x, y])
```

You can perform standard mathematical operations on points and vectors.

Perform some standard operations on vectors.

```
[2]: vec_1 = Vector3D([1,0,0]) # x-vector
      vec_2 = Vector3D([0,1,0]) # y-vector

      print("Sum of vectors [1, 0, 0] + [0, 1, 0]:")
      print(vec_1 + vec_2) # sum

      print("\nDot product of vectors [1, 0, 0] * [0, 1, 0]:")
      print(vec_1 * vec_2) # dot

      print("\nCross product of vectors [1, 0, 0] % [0, 1, 0]:")
      print(vec_1 % vec_2) # cross

Sum of vectors [1, 0, 0] + [0, 1, 0]:
[1 1 0]

Dot product of vectors [1, 0, 0] * [0, 1, 0]:
0

Cross product of vectors [1, 0, 0] % [0, 1, 0]:
[0 0 1]
```

Create a vector from two points.

```
[3]: p1 = Point3D([12.4, 532.3, 89])
      p2 = Point3D([-5.7, -67.4, 46.6])

      vec_3 = Vector3D.from_points(p1, p2)
      vec_3

[3]: Vector3D([ -18.1, -599.7, -42.4])
```

Normalize a vector to create a unit vector, which is also known as a *direction*.

```
[4]: print("Magnitude of vec_3:")
      print(vec_3.magnitude)

      print("\nNormalized vec_3:")
      print(vec_3.normalize())

      print("\nNew magnitude:")
      print(vec_3.normalize().magnitude)

Magnitude of vec_3:
601.4694173438911

Normalized vec_3:
[-0.03009297 -0.99705818 -0.07049402]
```

(continues on next page)

(continued from previous page)

```
New magnitude:
1.0
```

Use the `UnitVector` class to automatically normalize the input for the unit vector.

```
[5]: uv = UnitVector3D([1,1,1])
      uv
[5]: UnitVector3D([0.57735027, 0.57735027, 0.57735027])
```

Perform a few more mathematical operations on vectors.

```
[6]: v1 = Vector3D([1, 0, 0])
      v2 = Vector3D([0, 1, 0])

      print("Vectors are perpendicular:")
      print(v1.is_perpendicular_to(v2))

      print("\nVectors are parallel:")
      print(v1.is_parallel_to(v2))

      print("\nVectors are opposite:")
      print(v1.is_opposite(v2))

      print("\nAngle between vectors:")
      print(v1.get_angle_between(v2))
      print(f"{np.pi / 2} == pi/2")
```

```
Vectors are perpendicular:
True
```

```
Vectors are parallel:
False
```

```
Vectors are opposite:
False
```

```
Angle between vectors:
1.5707963267948966 radian
1.5707963267948966 == pi/2
```

Create planes

Once you begin creating sketches and bodies, Plane objects become very important. A plane is defined by these items:

- An origin, which consists of a 3D point
- Two directions (`direction_x` and `direction_y`), which are both `UnitVector3D` objects

If no direction vectors are provided, the plane defaults to the XY plane.

Create two planes.

```
[7]: plane = Plane(Point3D([0,0,0])) # XY plane
```

(continues on next page)

(continued from previous page)

```

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](#).

8.1.2 PyAnsys Geometry 101: Units

To handle units inside the source code, PyAnsys Geometry uses [Pint](#), a third-party open source software that other PyAnsys libraries also use.

The following code examples show how to operate with units inside the PyAnsys Geometry codebase and create objects with different units.

Import units handler

The following line of code imports the units handler: `pint.util.UnitRegistry`. For more information on the `UnitRegistry` class in the `pint` API, see [Most important classes](#) in the `Pint` documentation.

```
[1]: from ansys.geometry.core.misc import UNITS
```

Create and work with Quantity objects

With the `UnitRegistry` object called `UNITS`, you can create `Quantity` objects. A `Quantity` object is simply a container class with two core elements:

- A number
- A unit

`Quantity` objects have convenience methods, including those for transforming to different units and comparing magnitudes, values, and units. For more information on the `Quantity` class in the `pint` API, see [Most important classes](#) in the `Pint` documentation. You can also step through this [Pint tutorial](#).

```
[2]: from pint import Quantity

a = Quantity(10, UNITS.mm)

print(f"Object a is a pint.Quantity: {a}")

print("Request its magnitude in different ways (accessor methods):")
print(f"Magnitude: {a.m}.")
print(f"Also magnitude: {a.magnitude}.")

print("Request its units in different ways (accessor methods):")
print(f"Units: {a.u}.")
print(f"Also units: {a.units}.")
```

(continues on next page)

(continued from previous page)

```

# Quantities can be compared between different units
# You can also build Quantity objects as follows:
a2 = 10 * UNITS.mm
print(f"Compare quantities built differently: {a == a2}")

# Quantities can be compared between different units
a2_diff_units = 1 * UNITS.cm
print(f"Compare quantities with different units: {a == a2_diff_units}")

Object a is a pint.Quantity: 10 millimeter
Request its magnitude in different ways (accessor methods):
Magnitude: 10.
Also magnitude: 10.
Request its units in different ways (accessor methods):
Units: millimeter.
Also units: millimeter.
Compare quantities built differently: True
Compare quantities with different units: True

```

PyAnsys Geometry objects work by returning Quantity objects whenever the property requested has a physical meaning.

Return Quantity objects for Point3D objects.

```

[3]: from ansys.geometry.core.math import Point3D

point_a = Point3D([1,2,4])
print("===== Point3D([1,2,4]) =====")
print(f"Point3D is a numpy.ndarray in SI units: {point_a}.")
print(f"However, request each of the coordinates individually...\n")
print(f"X Coordinate: {point_a.x}")
print(f"Y Coordinate: {point_a.y}")
print(f"Z Coordinate: {point_a.z}\n")

# Now, store the information with different units...
point_a_km = Point3D([1,2,4], unit=UNITS.km)
print("===== Point3D([1,2,4], unit=UNITS.km) =====")
print(f"Point3D is a numpy.ndarray in SI units: {point_a_km}.")
print(f"However, request each of the coordinates individually...\n")
print(f"X Coordinate: {point_a_km.x}")
print(f"Y Coordinate: {point_a_km.y}")
print(f"Z Coordinate: {point_a_km.z}\n")

# These points, although they are in different units, can be added together.
res = point_a + point_a_km

print("===== res = point_a + point_a_km =====")
print(f"numpy.ndarray: {res}")
print(f"X Coordinate: {res.x}")
print(f"Y Coordinate: {res.y}")
print(f"Z Coordinate: {res.z}")

===== Point3D([1,2,4]) =====
Point3D is a numpy.ndarray in SI units: [1. 2. 4.].

```

(continues on next page)

(continued from previous page)

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
```

(continues on next page)

(continued from previous page)

```

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](#).

8.1.3 PyAnsys Geometry 101: Sketching

With PyAnsys Geometry, you can build powerful dynamic sketches without communicating with the Geometry service. This example shows how to build some simple sketches.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core.math import Plane, Point2D, Point3D, Vector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Add a box to sketch

The Sketch object is the starting point. Once it is created, you can dynamically add various curves to the sketch. Here are some of the curves that are available:

- arc
- box
- circle
- ellipse
- gear
- polygon
- segment
- slot
- trapezoid
- triangle

Add a box to the sketch.

```
[2]: sketch = Sketch()

sketch.segment(Point2D([0,0]), Point2D([0,1]))
sketch.segment(Point2D([0,1]), Point2D([1,1]))
sketch.segment(Point2D([1,1]), Point2D([1,0]))
sketch.segment(Point2D([1,0]), Point2D([0,0]))

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-
```

A *functional-style sketching API* is also implemented. It allows you to append curves to the sketch with the idea of *never picking up your pen*.

Use the functional-style sketching API to add a box.

```
[3]: sketch = Sketch()

(
    sketch.segment(Point2D([0,0]), Point2D([0,1]))
      .segment_to_point(Point2D([1,1]))
      .segment_to_point(Point2D([1,0]))
      .segment_to_point(Point2D([0,0]))
)

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-
```

A Sketch object uses the XY plane by default. You can define your own custom plane using three parameters: `origin`, `direction_x`, and `direction_y`.

Add a box on a custom plane.

```
[4]: plane = Plane(origin=Point3D([0,0,0]), direction_x=Vector3D([1,2,-1]), direction_
↪y=Vector3D([1,0,1]))
```

(continues on next page)

(continued from previous page)

```

sketch = Sketch(plane)

sketch.box(Point2D([0,0]), 1, 1)

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Combine concepts to create powerful sketches

Combine these simple concepts to create powerful sketches.

[5]: # *Complex Fluent API Sketch - PCB*

```

sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↳"LowerEdge")
    .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↳mm), tag="SupportedCorner")
    .segment_to_point(Point2D([41.5, 5], unit=UNITS.mm))
    .arc_to_point(Point2D([43, 6.5], unit=UNITS.mm), Point2D([43, 5], unit=UNITS.mm),
↳True)
    .segment_to_point(Point2D([55, 6.5], unit=UNITS.mm))
    .arc_to_point(Point2D([56.5, 8], unit=UNITS.mm), Point2D([55, 8], unit=UNITS.mm))
    .segment_to_point(Point2D([56.5, 35], unit=UNITS.mm))
    .arc_to_point(Point2D([55, 36.5], unit=UNITS.mm), Point2D([55, 35], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 36.5], unit=UNITS.mm))
    .segment_to_point(Point2D([0, 0], unit=UNITS.mm))
    .circle(Point2D([4, 4], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor1")
    .circle(Point2D([51, 34.5], UNITS.mm), Quantity(1.5, UNITS.mm), "Anchor2")
)

sketch.plot()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↳Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

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](#).

8.1.4 PyAnsys Geometry 101: Modeling

Once you understand PyAnsys Geometry's mathematical constructs, units, and sketching capabilities, you can dive into its modeling capabilities.

PyAnsys Geometry is a Python client that connects to a modeling service. Here are the modeling services that are available for connection:

- **DMS:** Windows-based modeling service that has been containerized to ease distribution, execution, and remotability operations.
- **Geometry service:** Linux-based approach of DMS that is currently under development.
- **Ansys Discovery and SpaceClaim:** PyAnsys Geometry is capable of connecting to a running session of Ansys Discovery or SpaceClaim. Although this is not the main use case for PyAnsys Geometry, a connection to one of these Ansys products is possible. Because these products have graphical user interfaces, performance is not as high with this option as with the previous options. However, connecting to a running instance of Discovery or SpaceClaim might be useful for some users.

Launch a modeling service

While the PyAnsys Geometry operations in earlier examples did not require communication with a modeling service, this example requires that a modeling service is available. All subsequent examples also require that a modeling service is available.

Launch a modeling service session.

```
[1]: from ansys.geometry.core import launch_modeler
```

```
# Start a modeler session
modeler = launch_modeler()
print(modeler)
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 700...
```

```
Ansys Geometry Modeler (0x7fb48b178440)
```

```
Ansys Geometry Modeler Client (0x7fb48b0b9400)
```

```
Target:      localhost:700
```

```
Connection: Healthy
```

```
Backend info:
```

```
Version:      27.1.0
```

```
Backend type: CORE_LINUX
```

```
Backend number: 20260130.2
```

```
API server number: 1984
```

```
CADIntegration: 27.1.0.7
```

```
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS.
↳on localhost:700. This is INSECURE. Consider using a secure connection.
```

(continues on next page)

(continued from previous page)

```
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↪ a secure connection.")
```

You can also launch your own services and connect to them. For information on connecting to an existing service, see the [Modeler API](#) documentation.

Here is how the class architecture is implemented:

- **Modeler:** Handler object for the active service session. This object allows you to connect to an existing service by passing in a host and a port. It also allows you to create **Design** objects, which is where the modeling takes place. For more information, see the [Modeler API](#) documentation.
- **Design:** Root object of your assembly (tree). While a **Design** object is also a **Component** object, it has enhanced capabilities, including creating named selections, adding materials, and handling beam profiles. For more information, see the [Design API](#) documentation.
- **Component:** One of the main objects for modeling purposes. **Component** objects allow you to create bodies, subcomponents, beams, design points, planar surfaces, and more. For more information, see the [Component API](#) documentation.

The following code examples show how you use these objects. More capabilities of these objects are shown in the specific example sections for sketching and modeling.

Create and plot a sketch

Create a **Sketch** object and plot it.

```
[2]: from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS, Distance

outer_hole_radius = Distance(0.5, UNITS.m)

sketch = Sketch()
(
    sketch.segment(start=Point2D([-4, 5], unit=UNITS.m), end=Point2D([4, 5], unit=UNITS.
↪ m))
    .segment_to_point(end=Point2D([4, -5], unit=UNITS.m))
    .segment_to_point(end=Point2D([-4, -5], unit=UNITS.m))
    .segment_to_point(end=Point2D([-4, 5], unit=UNITS.m))
    .box(
        center=Point2D([0, 0], unit=UNITS.m),
        width=Distance(3, UNITS.m),
        height=Distance(3, UNITS.m),
    )
    .circle(center=Point2D([3, 4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([-3, -4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([-3, 4], unit=UNITS.m), radius=outer_hole_radius)
    .circle(center=Point2D([3, -4], unit=UNITS.m), radius=outer_hole_radius)
)

# Plot the sketch
sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪ http-equiv=&quot;Content-...
```

Perform some modeling operations

Now that the sketch is ready to be extruded, perform some modeling operations, including creating the design, creating the body directly on the design, and plotting the body.

```
[3]: # Start by creating the Design
design = modeler.create_design("ModelingDemo")

# Create a body directly on the design by extruding the sketch
body = design.extrude_sketch(
    name="Design_Body", sketch=sketch, distance=Distance(80, unit=UNITS.cm)
)

# Plot the body
design.plot()
```

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_

↪http-equiv="Content-...

Perform some operations on the body

Perform some operations on the body.

```
[4]: # Request its faces, edges, volume...
faces = body.faces
edges = body.edges
volume = body.volume

print(f"This is body {body.name} with ID (server-side): {body.id}.")
print(f"This body has {len(faces)} faces and {len(edges)} edges.")
print(f"The body volume is {volume}.")
```

This is body Design_Body with ID (server-side): 0:22.
 This body has 14 faces and 32 edges.
 The body volume is 54.28672587712814 meter ** 3.

Other operations that can be performed include adding a midsurface offset and thickness (only for planar bodies), imprinting curves, assigning materials, copying, and translating.

Copy the body on a new subcomponent and translate it.

```
[5]: from ansys.geometry.core.math import UNITVECTOR3D_X

# Create a component
comp = design.add_component("Component")

# Copy the body that belongs to this new component
body_copy = body.copy(parent=comp, name="Design_Component_Body")

# Displace this new body by a certain distance (10m) in a certain direction (X-axis)
body_copy.translate(direction=UNITVECTOR3D_X, distance=Distance(10, unit=UNITS.m))

# Plot the result of the entire design
design.plot()
```

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_

↪http-equiv="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): 0:474.
```

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()

      WARNING - localhost:7000 - component - delete_body - Body 0:22 is not found in this
      ↪ component (or subcomponents). Ignoring this deletion request.

      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)
```

(continues on next page)

(continued from previous page)

```
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()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↳it was already running...
```

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](#).

8.1.5 PyAnsys Geometry 101: Plotter

This example provides an overview of PyAnsys Geometry's plotting capabilities, focusing on its plotter features. After reviewing the fundamental concepts of sketching and modeling in PyAnsys Geometry, it shows how to leverage these key plotting capabilities:

- **Multi-object plotting:** You can conveniently plot a list of elements, including objects created in both PyAnsys Geometry and PyVista libraries.
- **Interactive object selection:** You can interactively select PyAnsys Geometry objects within the scene. This enables efficient manipulation of these objects in subsequent scripting.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity
import pyvista as pv

from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import GeometryPlotter
from ansys.geometry.core.sketch import Sketch
```

Launch modeling service

Launch a modeling service session.

```
[2]: from ansys.geometry.core import launch_modeler

# Start a modeler session
modeler = launch_modeler()
print(modeler)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳ port 7000...

Ansys Geometry Modeler (0x7f54512d0c20)

Ansys Geometry Modeler Client (0x7f54512d16a0)
  Target:      localhost:7000
  Connection: Healthy
  Backend info:
    Version:      27.1.0
    Backend type:  CORE_LINUX
    Backend number: 20260130.2
    API server number: 1984
    CADIntegration: 27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳ on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳ a secure connection.")
```

You can also launch your own services and connect to them. For information on connecting to an existing service, see the [Modeler API](#) documentation.

Instantiate design and initialize object list

Instantiate a new design to work on and initialize a list of objects for plotting.

```
[3]: # init modeler
design = modeler.create_design("Multiplot")

plot_list = []
```

You are now ready to create some objects and use the plotter capabilities.

Create a PyAnsys Geometry body cylinder

Use PyAnsys Geometry to create a body cylinder.

```
[4]: cylinder = Sketch()
cylinder.circle(Point2D([10, 10], UNITS.m), 1.0)
cylinder_body = design.extrude_sketch("JustACyl", cylinder, Quantity(10, UNITS.m))
plot_list.append(cylinder_body)
```

Create a PyAnsys Geometry arc sketch

Use PyAnsys Geometry to create an arc sketch.

```
[5]: sketch = Sketch()
sketch.arc(
    Point2D([20, 20], UNITS.m),
    Point2D([20, -20], UNITS.m),
    Point2D([10, 0], UNITS.m),
    tag="Arc",
)
plot_list.append(sketch)
```

Create a PyVista cylinder

Use PyVista to create a cylinder.

```
[6]: cyl = pv.Cylinder(radius=5, height=20, center=(-20, 10, 10))
plot_list.append(cyl)
```

Create a PyVista multiblock

Use PyVista to create a multiblock with a sphere and a cube.

```
[7]: blocks = pv.MultiBlock(
    [pv.Sphere(center=(20, 10, -10), radius=10), pv.Cube(x_length=10, y_length=10, z_
    ↪length=10)]
)
plot_list.append(blocks)
```

Create a PyAnsys Geometry body box

Use PyAnsys Geometry to create a body box that is a cube.

```
[8]: box2 = Sketch()
box2.box(Point2D([-10, 20], UNITS.m), Quantity(10, UNITS.m), Quantity(10, UNITS.m))
box_body2 = design.extrude_sketch("JustABox", box2, Quantity(10, UNITS.m))
plot_list.append(box_body2)
```

Plot objects

When plotting the created objects, you have several options.

You can simply plot one of the created objects.


```
[9]: plotter = GeometryPlotter()
plotter.show(box_body2)

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

You can plot the whole list of objects.

```
[10]: plotter = GeometryPlotter()
plotter.show(plot_list)

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↪ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↪Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↪'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

The Python visualizer is used by default. However, you can also use `trame` for visualization.

```
plotter = GeometryPlotter(use_trame=True)
plotter.show(plot_list)
```

Clip objects

You can clip any object represented in the plotter by defining a Plane object that intersects the target object.

```
[11]: from ansys.geometry.core.math import Plane, Point3D
pl = GeometryPlotter()

# Define PyAnsys Geometry box
box2 = Sketch()
box2.box(Point2D([-10, 20], UNITS.m), Quantity(10, UNITS.m), Quantity(10, UNITS.m))
box_body2 = design.extrude_sketch("JustABox", box2, Quantity(10, UNITS.m))

# Define plane to clip the box
origin = Point3D([-10., 20., 5.], UNITS.m)
plane = Plane(origin=origin, direction_x=[1, 1, 1], direction_y=[-1, 0, 1])

# Add the object with the clipping plane
pl.plot(box_body2, clipping_plane=plane)
pl.show()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Select objects interactively

PyAnsys Geometry's plotter supports interactive object selection within the scene. This enables you to pick objects for subsequent script manipulation.

```
[12]: plotter = GeometryPlotter(allow_picking=True)

# Plotter returns picked bodies
picked_list = plotter.show(plot_list)
print(picked_list)

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv="Content-...'

None
```

It is also possible to enable picking directly for a specific design or component object alone. In the following cell, picking is enabled for the design object.

```
[13]: picked_list = design.plot(allow_picking=True)
print(picked_list)

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv="Content-...'

None
```

Render in different colors

You can render the objects in different colors automatically using PyVista's default color cycler. In order to do this, activate the `multi_colors=True` option when calling the `plot()` method.

In the following cell you can create a new design and plot a prism and a cylinder in different colors.

```
[14]: design = modeler.create_design("MultiColors")

# Create a sketch of a box
sketch_box = Sketch().box(Point2D([0, 0], unit=UNITS.m), width=30 * UNITS.m, height=40 *
↳ UNITS.m)

# Create a sketch of a circle (overlapping the box slightly)
sketch_circle = Sketch().circle(Point2D([20, 0], unit=UNITS.m), radius=3 * UNITS.m)

# Extrude both sketches to get a prism and a cylinder
design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

# Design plotting
design.plot(multi_colors=True)

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳ creating a new one.
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[15]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↳it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.2 Sketching examples

These examples demonstrate math operations on geometric objects and sketching capabilities, combined with server-based operations.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.2.1 Sketching: Basic usage

This example shows how to use basic PyAnsys Geometry sketching capabilities.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core.misc.units import UNITS as u
from ansys.geometry.core.sketch import Sketch
```

Create a sketch

Sketches are fundamental objects for drawing basic shapes like lines, segments, circles, ellipses, arcs, and polygons.

You create a `Sketch` instance by defining a drawing plane. To define a plane, you declare a point and two fundamental orthogonal directions.

```
[2]: from ansys.geometry.core.math import Plane, Point2D, Point3D
```

Define a plane for creating a sketch.

```
[3]: # Define the origin point of the plane
origin = Point3D([1, 1, 1])

# Create a plane located in previous point with desired fundamental directions
plane = Plane(
    origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
)

# Instantiate a new sketch object from previous plane
sketch = Sketch(plane)
```

Draw shapes

To draw different shapes in the sketch, you use draw methods.

Draw a circle

You draw a circle in a sketch by specifying the center and radius.

```
[4]: sketch.circle(Point2D([2, 1]), radius=30 * u.cm, tag="Circle")
sketch.select("Circle")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Draw an ellipse

You draw an ellipse in a sketch by specifying the center, major radius, and minor radius.

```
[5]: sketch.ellipse(
    Point2D([1, 1]), major_radius=2*u.m, minor_radius=1*u.m, tag="Ellipse"
)
sketch.select("Ellipse")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Draw a polygon

You draw a regular polygon by specifying the center, radius, and desired number of sides.

```
[6]: sketch.polygon(
    Point2D([1, 1]), inner_radius=3*u.m, sides=5, tag="Polygon"
)
sketch.select("Polygon")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Draw an arc

You draw an arc of circumference by specifying the center, starting point, and ending point.

```
[7]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
      sketch.arc(start_point, end_point, Point2D([1,1]), tag="Arc")
      sketch.select("Arc")
      sketch.plot_selection()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:266: PyVistaDeprecationWarning:
../.../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:266:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    arc_sub1 = pv.CircularArc(
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:271: PyVistaDeprecationWarning:
../.../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:271:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    arc_sub2 = pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv="Content-...
```

There are also additional ways to draw arcs, such as by specifying the start, center point, and angle.

```
[8]: start_point = Point2D([2, 1], unit=u.m)
      center_point = Point2D([1, 1], unit=u.m)
      angle = 90
      sketch.arc_from_start_center_and_angle(
          start_point, center_point, angle=90, tag="Arc_from_start_center_angle"
      )
      sketch.select("Arc_from_start_center_angle")
      sketch.plot_selection()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../.../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv="Content-...
```

Or by specifying the start, end point, and radius.

```
[9]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
      radius = 1 * u.m
      sketch.arc_from_start_end_and_radius(
          start_point, end_point, radius, tag="Arc_from_start_end_radius"
      )
```

(continues on next page)

(continued from previous page)

```

sketch.select("Arc_from_start_end_radius")
sketch.plot_selection()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:266: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:266:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    arc_sub1 = pv.CircularArc(
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:271: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:271:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    arc_sub2 = pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...
```

Draw a slot

You draw a slot by specifying the center, width, and height.

```

[10]: sketch.slot(Point2D([2, 0]), 4, 3, tag="Slot")
sketch.select("Slot")
sketch.plot_selection()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:266: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:266:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    arc_sub1 = pv.CircularArc(
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:271: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:271:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    arc_sub2 = pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...
```

Draw a box

You draw a box by specifying the center, width, and height.

```

[11]: sketch.box(Point2D([2, 0]), 4, 5, tag="Box")
sketch.select("Box")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Draw a segment

You draw a segment by specifying the starting point and ending point.

```
[12]: start_point, end_point = Point2D([2, 1], unit=u.m), Point2D([0, 1], unit=u.meter)
sketch.segment(start_point, end_point, "Segment")
sketch.select("Segment")
sketch.plot_selection()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Plot the sketch

The `Plotter` class provides capabilities for plotting different PyAnsys Geometry objects. PyAnsys Geometry uses PyVista as the visualization backend.

You use the `plot_sketch` method to plot a sketch. This method accepts a `Sketch` instance and some extra arguments to further customize the visualization of the sketch. These arguments include showing the plane of the sketch and its frame.

```
[13]: # Plot the sketch in the whole scene
from ansys.geometry.core.plotting import GeometryPlotter

pl = GeometryPlotter()
pl.add_sketch(sketch, show_plane=True, show_frame=True)
pl.show()
```

```
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/geometry/core/sketch/arc.py:266: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:266:
↳Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳'CircularArc'.
```

From version 0.50, passing these as positional arguments will result in a `TypeError`.

```
arc_sub1 = pv.CircularArc(
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/geometry/core/sketch/arc.py:271: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:271:
↳Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳'CircularArc'.
```

From version 0.50, passing these as positional arguments will result in a `TypeError`.

```
arc_sub2 = pv.CircularArc(
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↳Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳'CircularArc'.
```

From version 0.50, passing these as positional arguments will result in a `TypeError`.

```
return pv.CircularArc(
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](#).

8.2.2 Sketching: Dynamic sketch plane

The sketch is a lightweight, two-dimensional modeler driven primarily by client-side execution.

At any point, the current state of a sketch can be used for operations such as extruding a body, projecting a profile, or imprinting curves.

The sketch is designed as an effective *functional-style* API with all operations receiving 2D configurations.

For easy reuse of sketches across different regions of your design, you can move a sketch around the global coordinate system by modifying the plane defining the current sketch location.

This example creates a multi-layer PCB from many extrusions of the same sketch, creating unique design bodies for each layer.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_Z, Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile

You can create, modify, and plot `Sketch` instances independent of supporting Geometry service instances.

To define the sketch profile for the PCB, you create a sketch outline of individual `Segment` and `Arc` objects with two circular through-hole attachment points added within the profile boundary to maintain a single, closed sketch face.

Create a single `Sketch` instance to use for multiple design operations.

```
[2]: sketch = Sketch()

(
    sketch.segment(Point2D([0, 0], unit=UNITS.mm), Point2D([40, 1], unit=UNITS.mm),
↳ "LowerEdge")
    .arc_to_point(Point2D([41.5, 2.5], unit=UNITS.mm), Point2D([40, 2.5], unit=UNITS.
↳ mm), tag="SupportedCorner")
```

(continues on next page)

(continued from previous page)

```

        .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()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↪ ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↪ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↪ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪ http-equiv=&quot;Content-...
```

Instantiate the modeler

Launch a modeling service and connect to it.

```

[3]: modeler = launch_modeler()
    print(modeler)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↪ port 700...

Ansys Geometry Modeler (0x7f076d6727b0)

Ansys Geometry Modeler Client (0x7f076d672900)
  Target:      localhost:700
  Connection: Healthy
  Backend info:
    Version:      27.1.0
    Backend type:  CORE_LINUX
    Backend number: 20260130.2
    API server number: 1984
    CADIntegration: 27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↪ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↪ on localhost:700. This is INSECURE. Consider using a secure connection.
    warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↪ a secure connection.")
```

Extrude multiple bodies

Use the single sketch profile to extrude the board profile at multiple Z-offsets. Create a named selection from the resulting list of layer bodies.

Note that translating the sketch plane prior to extrusion is more effective (10 server calls) than creating a design body on the supporting server and then translating the body on the server (20 server calls).

```
[4]: design = modeler.create_design("ExtrudedBoardProfile")

layers = []
layer_thickness = Quantity(0.20, UNITS.mm)
for layer_index in range(10):
    layers.append(design.extrude_sketch(f"BoardLayer_{layer_index}", sketch, layer_
    ↪thickness))
    sketch.translate_sketch_plane_by_distance(UNITVECTOR3D_Z, layer_thickness)

board_named_selection = design.create_named_selection("FullBoard", bodies=layers)
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
    ↪http-equiv=&quot;Content-...
```

Close the modeler

Close the modeler to release the resources.

```
[5]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
    ↪it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.2.3 Sketching: Parametric sketching for gears

This example shows how to use gear sketching shapes from PyAnsys Geometry.

Perform required imports and pre-sketching operations

Perform required imports and instantiate the `Modeler` instance and the basic elements that define a sketch.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point2D, Point3D
from ansys.geometry.core.misc import UNITS, Distance
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.plotting import GeometryPlotter

# Start a modeler session
modeler = launch_modeler()
print(modeler)

# Define the origin point of the plane
origin = Point3D([1, 1, 1])

# Create a plane containing the previous point with desired fundamental directions
plane = Plane(
    origin, direction_x=[1, 0, 0], direction_y=[0, -1, 1]
)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 700...

Ansys Geometry Modeler (0x7f35a4364c20)

Ansys Geometry Modeler Client (0x7f35a43656a0)
Target:      localhost:700
Connection: Healthy
Backend info:
  Version:           27.1.0
  Backend type:      CORE_LINUX
  Backend number:    20260130.2
  API server number: 1984
  CADIntegration:    27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")
```

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)
```

(continues on next page)

(continued from previous page)

```

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()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...
```

After creating the sketch, extrudes it.

```

[3]: # Create a design
design = modeler.create_design("AdvancedFeatures_DummyGear")

# Extrude your sketch
dummy_gear = design.extrude_sketch("DummyGear", sketch, Distance(1000, UNITS.mm))

# Plot the design
design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...
```

Sketch a spur gear

SpurGear sketches are parametric CAD spur gears based on four parameters:

- **origin**: Center point location for the desired spur gear. The value must be a `Point2D` object.
- **module**: Ratio between the pitch circle diameter in millimeters and the number of teeth. This is a common parameter for spur gears. The value should be an integer or a float.
- **pressure_angle**: Pressure angle expected for the teeth of the spur gear. This is also a common parameter for spur gears. The value must be a `pint.Quantity` object.
- **n_teeth**: Number of teeth. The value must be an integer.

Instantiate a new `Sketch` object and then define and plot a spur gear.

```

[4]: # Instantiate a new sketch object from previous plane
sketch = Sketch(plane)

# Define spur gear
#
origin = Point2D([0, 1], unit=UNITS.meter)
module = 40
```

(continues on next page)

(continued from previous page)

```

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()

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
↳ Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
↳ 'CircularArc'.
From version 0.50, passing these as positional arguments will result in a TypeError.
    return pv.CircularArc(

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...
```

After creating the sketch, extrude it.

```

[5]: # Create a design
design = modeler.create_design("AdvancedFeatures_SpurGear")

# Extrude sketch
dummy_gear = design.extrude_sketch("SpurGear", sketch, Distance(200, UNITS.mm))

# Plot design
design.plot()

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳ creating a new one.

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...
```

Close the modeler

```

[6]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↳ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3 Modeling examples

These examples demonstrate service-based modeling operations.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.1 Modeling: Single body with material assignment

In PyAnsys Geometry, a *body* represents solids or surfaces organized within the Design assembly. The current state of sketch, which is a client-side execution, can be used for the operations of the geometric design assembly.

The Geometry service provides data structures to create individual materials and their properties. These data structures are exposed through PyAnsys Geometry.

This example shows how to create a single body from a sketch by requesting its extrusion. It then shows how to assign a material to this body.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.materials import Material, MaterialProperty, \
    ↳MaterialPropertyType
from ansys.geometry.core.math import UNITVECTOR3D_Z, Frame, Plane, Point2D, Point3D, \
    ↳UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Create sketch

Create a Sketch instance and insert a circle with a radius of 10 millimeters in the default plane.

```
[2]: sketch = Sketch()
sketch.circle(Point2D([10, 10], UNITS.mm), Quantity(10, UNITS.mm))

[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x7fb11bf3cc20>
```

Initiate design on server

Launch a modeling service session and initiate a design on the server.

```
[3]: # Start a modeler session
modeler = launch_modeler()
print(modeler)

design_name = "ExtrudeProfile"
design = modeler.create_design(design_name)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 7000...

Ansys Geometry Modeler (0x7fb11bf3ee40)

Ansys Geometry Modeler Client (0x7fb11be263c0)
Target:      localhost:7000
Connection: Healthy
Backend info:
  Version:      27.1.0
  Backend type:  CORE_LINUX
  Backend number: 20260130.2
  API server number: 1984
  CADIntegration: 27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")
```

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)
```

(continues on next page)

(continued from previous page)

```
body.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[6]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↪it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.2 Modeling: Rectangular plate with multiple bodies

You can create multiple bodies from a single sketch by extruding the same sketch in different planes.

The sketch is designed as an effective *functional-style* API with all operations receiving 2D configurations. For more information, see the *Sketch* user guide.

In this example, a box is located in the center of the plate, with the default origin of a sketch plane (origin at (0, 0, 0)). Four holes of equal radius are sketched at the corners of the plate. The plate is then extruded, leading to the generation of the requested body. The projection is at the center of the face. The default projection depth is through the entire part.

Perform required imports

Perform the required imports.

```
[1]: import numpy as np
from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point3D, Point2D, UnitVector3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```


Define sketch profile

The sketch profile for the proposed design requires four segments that constitute the outer limits of the design, a box on the center, and a circle at its four corners.

You can use a single sketch instance for multiple design operations, including extruding a body, projecting a profile, and imprinting curves.

Define the sketch profile for the rectangular plate with multiple bodies.

```
[2]: sketch = Sketch()
(sketch.segment(Point2D([-4, 5], unit=UNITS.m), Point2D([4, 5], unit=UNITS.m))
 .segment_to_point(Point2D([4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, 5], unit=UNITS.m))
 .box(Point2D([0,0], unit=UNITS.m), Quantity(3, UNITS.m), Quantity(3, UNITS.m))
 .circle(Point2D([3, 4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 .circle(Point2D([-3, -4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 .circle(Point2D([-3, 4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 .circle(Point2D([3, -4], unit=UNITS.m), Quantity(0.5, UNITS.m))
 )
```

```
[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x7f6bfa468ad0>
```

Extrude sketch to create design

Establish a server connection and use the single sketch profile to extrude the base component at the Z axis. Create a named selection from the resulting list of bodies. In only three server calls, the design extrudes the four segments with the desired thickness.

```
[3]: modeler = launch_modeler()

design = modeler.create_design("ExtrudedPlate")

body = design.extrude_sketch(f"PlateLayer", sketch, Quantity(2, UNITS.m))

board_named_selection = design.create_named_selection("Plate", bodies=[body])
design.plot()
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
↳ port 7000...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳ on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳ a secure connection.")
```

```
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=&quot;Content-...
```

Close the modeler

Close the modeler to free up resources and release the connection.

```
[7]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↳it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.3 Modeling: Extruded plate with cut operations

As seen in the *Rectangular plate with multiple bodies* example, you can create a complex sketch with holes and extrude it to create a body. However, you can also perform cut operations on the extruded body to achieve similar results.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point3D, Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Define sketch profile without holes

Create a sketch profile for the proposed design. The sketch is the same as the *Rectangular plate with multiple bodies* example, but without the holes.

These holes are created by performing cut operations on the extruded body in the next steps.

```
[2]: sketch = Sketch()
(sketch.segment(Point2D([-4, 5], unit=UNITS.m), Point2D([4, 5], unit=UNITS.m))
 .segment_to_point(Point2D([4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, -5], unit=UNITS.m))
 .segment_to_point(Point2D([-4, 5], unit=UNITS.m))
 .box(Point2D([0,0], unit=UNITS.m), Quantity(3, UNITS.m), Quantity(3, UNITS.m))
)

modeler = launch_modeler()
design = modeler.create_design("ExtrudedPlateNoHoles")
body = design.extrude_sketch(f"PlateLayer", sketch, Quantity(2, UNITS.m))

design.plot()
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
port 700...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
on localhost:700. This is INSECURE. Consider using a secure connection.
```

(continues on next page)

(continued from previous page)

```
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↪ a secure connection.")

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪ http-equiv="Content-
```

Define sketch profile for holes

Create a sketch profile for the holes in the proposed design. The holes are created by sketching circles at the four corners of the plate. First create a reference sketch for all the circles. This sketch is translated to the four corners of the plate.

```
[3]: sketch_hole = Sketch()
sketch_hole.circle(Point2D([0, 0], unit=UNITS.m), Quantity(0.5, UNITS.m))

hole_centers = [
    Plane(Point3D([3, 4, 0], unit=UNITS.m)),
    Plane(Point3D([-3, 4, 0], unit=UNITS.m)),
    Plane(Point3D([-3, -4, 0], unit=UNITS.m)),
    Plane(Point3D([3, -4, 0], unit=UNITS.m)),
]
```

Perform cut operations on the extruded body

Perform cut operations on the extruded body to create holes at the four corners of the plate.

```
[4]: for center in hole_centers:
    sketch_hole.plane = center
    design.extrude_sketch(
        name= f"H_{center.origin.x}_{center.origin.y}",
        sketch=sketch_hole,
        distance=Quantity(2, UNITS.m),
        cut=True,
    )

design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪ http-equiv="Content-
```

Close the modeler

Close the modeler to free up resources and release the connection.

```
[5]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↪ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.4 Modeling: Tessellation of two bodies

This example shows how to create two stacked bodies and return the tessellation as two merged bodies.

Perform required imports

Perform the required imports.

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D, Point3D, Plane
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Create design

Create the basic sketches to be tessellated and extrude the sketch in the required plane. For more information on creating a component and extruding a sketch in the design, see the [Rectangular plate with multiple bodies](#) example.

Here is a typical situation in which two bodies, with different sketch planes, merge each body into a single dataset. This effectively combines all the faces of each individual body into a single dataset without separating faces.

```
[2]: modeler = launch_modeler()

sketch_1 = Sketch()
box = sketch_1.box(
    Point2D([10, 10], unit=UNITS.m), width=Quantity(10, UNITS.m), height=Quantity(5,
    UNITS.m)
)
circle = sketch_1.circle(
    Point2D([0, 0], unit=UNITS.m), radius=Quantity(25, UNITS.m)
)

design = modeler.create_design("TessellationDesign")
comp = design.add_component("TessellationComponent")
body = comp.extrude_sketch("Body", sketch=sketch_1, distance=10 * UNITS.m)

# Create the second body in a plane with a different origin
sketch_2 = Sketch(Plane([0, 0, 10]))
box = sketch_2.box(Point2D(
    [10, 10], unit=UNITS.m), width=Quantity(10, UNITS.m), height=Quantity(5, UNITS.m)
)
circle = sketch_2.circle(
    Point2D([0, 10], unit=UNITS.m), radius=Quantity(25, UNITS.m)
)
```

(continues on next page)

(continued from previous page)

```
body = comp.extrude_sketch("Body", sketch=sketch_2, distance=10 * UNITS.m)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 700...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")
```

Tessellate component as two merged bodies

Tessellate the component and merge each body into a single dataset. This effectively combines all the faces of each individual body into a single dataset without separating faces.

```
[3]: dataset = comp.tessellate()
dataset

[3]: PolyData (0x7f7e1c375900)
  N Cells:    976
  N Points:   996
  N Strips:    0
  X Bounds:  -2.500e+01, 2.500e+01
  Y Bounds:  -2.499e+01, 3.499e+01
  Z Bounds:   0.000e+00, 2.000e+01
  N Arrays:    0
```

Single body tessellation is possible. In that case, users can request the body-level tessellation method to tessellate the body and merge all the faces into a single dataset.

```
[4]: dataset = comp.bodies[0].tessellate()
dataset

[4]: MultiBlock (0x7f7e1c3759c0)
  N Blocks:    7
  X Bounds:  -2.500e+01, 2.500e+01
  Y Bounds:  -2.499e+01, 2.499e+01
  Z Bounds:   0.000e+00, 1.000e+01
```

Plot design

Plot the design.

```
[5]: design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳http-equiv="Content-...
```

Close the modeler

Close the modeler.

```
[6]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↳ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.5 Modeling: Design organization

The Design instance creates a design project within the remote Geometry service to complete all CAD modeling against.

You can organize all solid and surface bodies in each design within a customizable component hierarchy. A component is simply an organization mechanism.

The top-level design node and each child component node can have one or more bodies assigned and one or more components assigned.

The API requires each component of the design hierarchy to be given a user-defined name.

There are several design operations that result in a body being created within a design. Executing each of these methods against a specific component instance explicitly specifies the node of the design tree to place the new body under.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, Point2D
from ansys.geometry.core.misc import UNITS, Distance
from ansys.geometry.core.sketch import Sketch
```

Organize design

Extrude two sketches to create bodies. Assign the cylinder to the top-level design component. Assign the slot to the component nested one level beneath the top-level design component.

```
[2]: modeler = launch_modeler()

design = modeler.create_design("DesignHierarchyExample")

circle_sketch = Sketch()
circle_sketch.circle(Point2D([10, 10], UNITS.mm), Distance(10, UNITS.mm))

cylinder_body = design.extrude_sketch("10mmCylinder", circle_sketch, Distance(10, UNITS.
```

(continues on next page)

(continued from previous page)

```

    ↪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()

```

WARNING - - docker_instance - _check_port_availability - Service is already running at
 ↪port 7000...

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
 ↪ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
 ↪on localhost:7000. This is INSECURE. Consider using a secure connection.
 warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
 ↪a secure connection.")

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
 ↪http-equiv="Content-...

Create nested component

Create a component that is nested under the previously created component and then create another cylinder from the previously used sketch.

```

[3]: double_nested_component = nested_component.add_component("DoubleNestedComponent")

circle_surface_body = double_nested_component.create_surface("CircularSurfaceBody",
    ↪circle_sketch)
circle_surface_body.translate(UNITVECTOR3D_X, Distance(-35, UNITS.mm))

design.plot()

```

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta
 ↪http-equiv="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="Content-...

Close the modeler

Close the modeler to release the resources.

```
[5]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↳ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.6 Modeling: Boolean operations

This example shows how to use Boolean operations for geometry manipulation.

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.designer import Body
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.plotting import GeometryPlotter
from ansys.geometry.core.sketch import Sketch
```

Launch local modeler

Launch the local modeler. If you are not familiar with how to launch the local modeler, see the “Launch a modeling service” section in the *PyAnsys Geometry 101: Modeling* example.

```
[2]: modeler = launch_modeler()
print(modeler)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳ port 7000...

Ansys Geometry Modeler (0x7eff82c0cec0)

Ansys Geometry Modeler Client (0x7eff82c0d940)
  Target:      localhost:7000
  Connection: Healthy
  Backend info:
```

(continues on next page)

(continued from previous page)

```

Version:          27.1.0
Backend type:     CORE_LINUX
Backend number:   20260130.2
API server number: 1984
CADIntegration:   27.1.0.7

```

```

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS.
↳ on localhost:7000. This is INSECURE. Consider using a secure connection.
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using.
↳ a secure connection.")

```

Define bodies

This section defines the bodies to use the Boolean operations on. First you create sketches of a box and a circle, and then you extrude these sketches to create 3D objects.

Create sketches

Create sketches of a box and a circle that serve as the basis for your bodies.

```

[3]: # Create a sketch of a box
sketch_box = Sketch().box(Point2D([0, 0], unit=UNITS.m), width=30 * UNITS.m, height=40 *
↳ UNITS.m)

# Create a sketch of a circle (overlapping the box slightly)
sketch_circle = Sketch().circle(Point2D([20, 0], unit=UNITS.m), radius=10 * UNITS.m)

```

Extrude sketches

After the sketches are created, extrude them to create 3D objects.

```

[4]: # Create a design
design = modeler.create_design("example_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

```

You must extrude the sketches each time that you perform an example operation. This is because performing a Boolean operation modifies the underlying design permanently. Thus, you no longer have two bodies. As shown in the Boolean operations themselves, whenever you pass in a body, it is consumed, and so it no longer exists. The remaining body (with the performed Boolean operation) is the one that performed the call to the method.

Select bodies

You can optionally select bodies in the plotter as described in the “Select objects interactively” section in the *PyAnsys Geometry 101: Plotter* example. As shown in this example, the plotter preserves the picking order, meaning that the output list is sorted according to the picking order.

```

pl = GeometryPlotter(allow_picking=True)
pl.plot(design.bodies)

```

(continues on next page)

(continued from previous page)

```
pl.show()
bodies: list[Body] = GeometryPlotter(allow_picking=True).show(design.bodies)
```

Otherwise, you can select bodies from the design directly.

```
[5]: bodies = [design.bodies[0], design.bodies[1]]
```

Perform Boolean operations

This section performs Boolean operations on the defined bodies using the PyAnsys Geometry library. It explores intersection, union, and subtraction operations.

Perform an intersection operation

To perform an intersection operation on the bodies, first set up the bodies.

```
[6]: # Create a design
design = modeler.create_design("intersection_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

```
WARNING - localhost:7000 - modeler - create_design - Closing previous design before
creating a new one.
```

Perform the intersection and plot the results.

```
[7]: prism.intersect(cylin)
_ = GeometryPlotter().show(design.bodies)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
http-equiv="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 0x7eff82aaea50
    Name          : Prism
    Exists        : True
    Parent component : intersection_design
    MasterBody     : 0:22
    Surface body   : False
    Color          : #D6F7D1
]
```

Perform a union operation

To carry out a union operation on the bodies, first set up the bodies.

```
[9]: # Create a design
design = modeler.create_design("union_design")
```

(continues on next page)

(continued from previous page)

```
# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

```
WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳creating a new one.
```

Perform the union and plot the results.

```
[10]: prism.unite(cylin)
_ = GeometryPlotter().show(design.bodies)

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳http-equiv="Content-...
```

The final remaining body is the prism body because the cylin body has been consumed.

```
[11]: print(design.bodies)

[
  ansys.geometry.core.designer.Body 0x7eff78880910
    Name          : Prism
    Exists        : True
    Parent component : union_design
    MasterBody     : 0:22
    Surface body   : False
    Color         : #D6F7D1
]
```

Perform a subtraction operation

To perform a subtraction operation on the bodies, first set up the bodies.

```
[12]: # Create a design
design = modeler.create_design("subtraction_design")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)
```

```
WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳creating a new one.
```

Perform the subtraction and plot the results.

```
[13]: prism.subtract(cylin)
_ = GeometryPlotter().show(design.bodies)

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳http-equiv="Content-...
```

The final remaining body is the prism body because the cylin body has been consumed.

```
[14]: print(design.bodies)

[
  ansys.geometry.core.designer.Body 0x7eff7894f070
    Name          : Prism
    Exists         : True
    Parent component : subtraction_design
    MasterBody      : 0:22
    Surface body    : False
    Color           : #D6F7D1
]
```

If you perform this action inverting the order of the bodies (that is, `cylin.subtract(prism)`), you can see the difference in the resulting shape of the body.

```
[15]: # Create a design
design = modeler.create_design("subtraction_design_inverted")

# Extrude both sketches to get a prism and a cylinder
prism = design.extrude_sketch("Prism", sketch_box, 50 * UNITS.m)
cylin = design.extrude_sketch("Cylinder", sketch_circle, 50 * UNITS.m)

# Invert subtraction
cylin.subtract(prism)
_ = GeometryPlotter().show(design.bodies)

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
creating a new one.

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
http-equiv="Content-...
```

In this case, the final remaining body is the `cylin` body because the `prism` body has been consumed.

```
[16]: print(design.bodies)

[
  ansys.geometry.core.designer.Body 0x7eff78888dd0
    Name          : Cylinder
    Exists         : True
    Parent component : subtraction_design_inverted
    MasterBody      : 0:85
    Surface body    : False
    Color           : #D6F7D1
]
```

Close the modeler

Close the modeler to release the resources.

```
[17]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
it was already running...
```

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](#).

8.3.7 Modeling: Scale, map and mirror bodies

The purpose of this notebook is to demonstrate the `map()` and `scale()` functions and their usage for transforming bodies.

```
[1]: # Imports
import numpy as np

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
    Frame,
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)

WARNING - - docker_instance - _check_port_availability - Service is already running at_
↪port 700...

Ansys Geometry Modeler (0x7ffa73859160)

Ansys Geometry Modeler Client (0x7ffa7378d400)
Target:      localhost:700
```

(continues on next page)

(continued from previous page)

```

Connection: Healthy
Backend info:
  Version:          27.1.0
  Backend type:     CORE_LINUX
  Backend number:   20260130.2
  API server number: 1984
  CADIntegration:   27.1.0.7

```

```

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳ on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳ a secure connection.")

```

Scale body

The `scale()` function is designed to modify the size of 3D bodies by a specified scale factor. This function is an important part of geometric transformations, allowing for the dynamic resizing of bodies.

Usage of `scale()`

To use the `scale()` function, you call it on an instance of a geometry body, passing a single argument: the scale value. This value is a real number (Real) that determines the factor by which the body's size is changed.

```
body.scale(value)
```

Example: Making a cube

The following code snippets show how to change the size of a cube using the `scale()` function in Body objects. The process involves initializing a sketch design for the cube, defining the shape parameters, and then performing a rescaling operation to generate the new shape.

Initialize the cube sketch design

A new design sketch named “cube” is created.

```
[3]: design = m.create_design("cube")
```

Define cube parameters

`side_length` is set to 10 units, representing the side length of the cube.

```
[4]: # Cube parameters
side_length = 10
```

Create the profile cube

A square box is created centered on the origin using `side_length` as the side length of the square.

```
[5]: # Square with side length 10
box_sketch = Sketch().box(Point2D([0, 0]), side_length, side_length)
```

(continues on next page)

(continued from previous page)

```
box_sketch.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Create cube body

extrude_sketch on the box_sketch as the base sketch and create the 3D cube with distance being the side_length.

```
[6]: # Extrude the cube profile by a distance of side_length
cube = design.extrude_sketch("box", box_sketch, side_length)

design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Making the cube twice as large

- Copy the original cube. Using scale() with a value of 2, double the side lengths of the cube, thereby making the body twice as large, and then offset it to view the difference.

```
[7]: # Copy the original cube
doubled = cube.copy(cube.parent_component, "doubled_box")
# Double the size
doubled.scale(2)
# Translate the copied cube in the x direction
doubled.translate(UNITVECTOR3D_X, 30)

design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

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")

WARNING - localhost:7000 - modeler - create_design - Closing previous design before_
↪creating a new one.
```

Create an asymmetric sketch profile

Make a sketch profile that is basically a cube centered on the origin with a side length of 2 with a cutout.

```
[10]: # Create the cube profile with a cut through it
asymmetric_profile = Sketch()
(
    asymmetric_profile.segment(Point2D([1, 1]), Point2D([-1, 1]))
    .segment_to_point(Point2D([0, 0.5]))
    .segment_to_point(Point2D([-1, -1]))
    .segment_to_point(Point2D([1, -1]))
    .segment_to_point(Point2D([1, 1]))
)

asymmetric_profile.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Create the asymmetric body

extrude_sketch on the asymmetric_profile as the base sketch, creating the 3D cube with a cutout, with the distance being 1.

```
[11]: # Extrude the asymmetric profile by a distance of 1 unit
body = design.extrude_sketch("box", asymmetric_profile, 1)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv=&quot;Content-...
```

Apply map reframing

First make a copy of the shape and translate it in 3D space so that you can view them side by side. Then, apply the reframing to the copied shape.

Note: The following map uses the default x direction, but the y direction is swapped with the z direction, effectively rotating the original shape so that it is standing vertically.

```
[12]: # Copy the body
copied_body = body.copy(body.parent_component, "copied_body")
# Apply the reframing
copied_body.map(Frame(Point3D([0, 0, 0]), UNITVECTOR3D_X, UNITVECTOR3D_Z))
# Shift the new modified body in the plane in the negative y direction by 2 units
copied_body.translate(UNITVECTOR3D_Y, -2)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↵http-equiv=&quot;Content-...
```

Mirror body

The mirror() function is designed to mirror the geometry of a body across a specified plane. This function plays a role in geometric transformations, enabling the reflection of bodies to create symmetrical designs.

Usage of mirror()

To use the mirror() function, you call it on an instance of a geometry body, passing a single argument: the plane across which to mirror the body. This plane is represented by a Plane object, defining the axis of symmetry for the mirroring operation.

```
body.mirror(plane)
```

Example: Triangle body

The following code snippets show how to use the mirror() function to reframe a cube body in the Body object. The process involves initializing a sketch design for the body profile, extruding the profile by a distance, and then performing the mirroring operation to reflect the shape over the specified axis.

Initialize the shape sketch design

A new design sketch named “triangle” is created.

```
[13]: # Initialize the sketch design
design = m.create_design("triangle")

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳ creating a new one.
```

Define parameters

point1: First vertex of the triangle. point2: Second vertex of the triangle. point3: Third vertex of the triangle.

```
[14]: point1 = Point2D([5, 0])
point2 = Point2D([2.5, 2.5])
point3 = Point2D([2.5, -2.5])
```

Create triangle sketch profile

Using point1, point2, and point3, define the vertices of the triangle profile using those three points and then create line segments connecting them.

```
[15]: # Draw the triangle sketch profile
sketch = Sketch()
sketch.segment(start=point1, end=point2)
sketch.segment(start=point2, end=point3)
sketch.segment(start=point3, end=point1)

sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳ http-equiv=&quot;Content-...
```

Create triangular body

Using the sketch profile created in the previous step, use the `extrude_sketch` method to create a solid body with a depth of 1.

```
[16]: # Extrude the triangular body by a distance of 1
triangle = design.extrude_sketch("triangle_body", sketch, 1)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳ http-equiv=&quot;Content-...
```

Mirror the triangular body

First, make a copy of the triangular body. Then, using `mirror()`, you can mirror the copied body over the ZY plane.

```
[17]: # Copy triangular body
mirrored_triangle = triangle.copy(triangle.parent_component, "mirrored_triangle")
# Mirror the copied body over the ZY plane (specified by the (0, 1, 0) and
```

(continues on next page)

(continued from previous page)

```
# (0, 0, 1) unit vectors)

mirrored_triangle.mirror(Plane(direction_x=UNITVECTOR3D_Y, direction_y=UNITVECTOR3D_Z))

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Closing the modeler

```
[18]: m.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↪it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.8 Modeling: Sweep chain and sweep sketch

This example shows how use the `sweep_sketch()` and `sweep_chain()` functions to create more complex extrusion profiles. You use the `sweep_sketch()` function with a closed sketch profile and the `sweep_chain()` function for an open profile.

```
[1]: # Imports
import numpy as np

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.shapes import Circle, Ellipse, Interval
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

WARNING - - docker_instance - _check_port_availability - Service is already running at port 700...

Ansys Geometry Modeler (0x7f40dede9160)

Ansys Geometry Modeler Client (0x7f40ded1d400)

Target: localhost:700

Connection: Healthy

Backend info:

Version:	27.1.0
Backend type:	CORE_LINUX
Backend number:	20260130.2
API server number:	1984
CADIntegration:	27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS on localhost:700. This is INSECURE. Consider using a secure connection.

warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using a secure connection.")

Example: Creating a donut

The following code snippets show how to use the `sweep_sketch()` function to create a 3D donut shape in the Design object. The process involves initializing a sketch design for the donut, defining two circles for the profile and path, and then performing a sweep operation to generate the donut shape.

Initialize the donut sketch design

A new design sketch named “donut” is created.

```
[3]: # Initialize the donut sketch design
design_sketch = m.create_design("donut")
```

Define circle parameters

`path_radius` is set to 5 units, representing the radius of the circular path that the profile circle sweeps along. `profile_radius` is set to 2 units, representing the radius of the profile circle that sweeps along the path to create the donut body.

```
[4]: # Donut parameters
path_radius = 5
profile_radius = 2
```

Create the profile circle

A circle is created on the XZ-plane centered at the coordinates (5, 0, 0) with a radius defined by `profile_radius`. This circle serves as the profile or cross-sectional shape of the donut.

```
[5]: # Create the circular profile on the XZ plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(direction_x=UNITVECTOR3D_X, direction_y=UNITVECTOR3D_Z)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([path_radius, 0]), profile_radius)

profile.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Create the path circle

Another circle, representing the path along which the profile circle is swept, is created on the XY-plane centered at (0, 0, 0). The radius of this circle is defined by `path_radius`.

```
[6]: # Create the circular path on the XY plane centered at (0, 0, 0) with radius 5
path = [Circle(Point3D([0, 0, 0]), path_radius).trim(Interval(0, 2 * np.pi))]
```

Perform the sweep operation

The sweep operation uses the profile circle and sweeps it along the path circle to create the 3D body of the donut. The result is stored in the variable `body`.

```
[7]: # Perform the sweep and examine the final body created
body = design_sketch.sweep_sketch("donutsweep", profile, path)

design_sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Example: Creating a bowl

This code demonstrates the process of using the `sweep_chain()` function to create a 3D model of a stretched bowl in the Design object. The model is generated by defining a quarter-ellipse as a profile and sweeping it along a circular path, creating a bowl shape with a stretched profile.

Initialize the bowl design

A design chain named “bowl” is created to initiate the bowl design process.

```
[8]: # Initialize the bowl sketch design
design_chain = m.create_design("bowl")

WARNING - localhost:7000 - modeler - create_design - Closing previous design before_
↪creating a new one.
```

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("bowlsweep", path, profile)

design_chain.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Closing the modeler

```
[13]: # Close the modeler
m.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↳it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.9 Modeling: Revolving a sketch

This example shows how to use the `revolve_sketch()` method to revolve a sketch around an axis to create a 3D body. You can also specify the angle of revolution to create a partial body.

```
[1]: # Imports
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import (
    Plane,
    Point2D,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.misc import UNITS, Angle
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 700...
```

```
Ansys Geometry Modeler (0x7f42bb3b1160)
```

```
Ansys Geometry Modeler Client (0x7f42bb2e9400)
```

```
Target:      localhost:700
```

```
Connection: Healthy
```

```
Backend info:
```

```
Version:      27.1.0
```

```
Backend type: CORE_LINUX
```

```
Backend number: 20260130.2
```

```
API server number: 1984
```

```
CADIntegration: 27.1.0.7
```

```
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:700. This is INSECURE. Consider using a secure connection.
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")
```


Example: Creating a quarter of a donut

The following code snippets show how to use the `revolve_sketch()` function to create a quarter of a 3D donut. The process involves defining a quarter of a circle as a profile and then revolving it around the Z-axis to create a 3D body.

Initialize the sketch design

Create a design sketch named `quarter-donut`.

```
[3]: # Initialize the donut sketch design
design = m.create_design("quarter-donut")
```

Define circle parameters

Set `path_radius`, which represents the radius of the circular path that the profile circle sweeps along, to 5 units. Set `profile_radius`, which represents the radius of the profile circle that sweeps along the path to create the donut body, to 2 units.

```
[4]: # Donut parameters
path_radius = 5
profile_radius = 2
```

Create the profile circle

Create a circle on the XZ plane centered at the coordinates (5, 0, 0) and use `profile_radius` to define the radius. This circle serves as the profile or cross-sectional shape of the donut.

```
[5]: # Create the circular profile on the XZ-plane centered at (5, 0, 0)
# with a radius of 2
plane_profile = Plane(
    origin=Point3D([path_radius, 0, 0]),
    direction_x=UNITVECTOR3D_X,
    direction_y=UNITVECTOR3D_Z,
)
profile = Sketch(plane=plane_profile)
profile.circle(Point2D([0, 0]), profile_radius)

profile.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↩http-equiv=&quot;Content-...
```

Perform the revolve operation

Revolve the profile circle around the Z axis to create a quarter of a donut body. Set the angle of revolution to 90 degrees in the default direction, which is counterclockwise.

```
[6]: # Revolve the profile around the Z axis and center in the absolute origin
# for an angle of 90 degrees
design.revolve_sketch(
    "quarter-donut-body",
    sketch=profile,
    axis=UNITVECTOR3D_Z,
```

(continues on next page)

(continued from previous page)

```

    angle=Angle(90, unit=UNITS.degrees),
    rotation_origin=Point3D([0, 0, 0]),
)

design.plot()

```

```

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...

```

Perform a revolve operation with a negative angle of revolution

You can use a negative angle of revolution to create a quarter of a donut in the opposite direction. The following code snippet shows how to create a quarter of a donut in the clockwise direction. The same profile circle is used, but the angle of revolution is set to -90 degrees.

```

[7]: # Initialize the donut sketch design
design = m.create_design("quarter-donut-negative")

# Revolve the profile around the Z axis and center in the absolute origin
# for an angle of -90 degrees (clockwise)
design.revolve_sketch(
    "quarter-donut-body-negative",
    sketch=profile,
    axis=UNITVECTOR3D_Z,
    angle=Angle(-90, unit=UNITS.degrees),
    rotation_origin=Point3D([0, 0, 0]),
)

design.plot()

```

```

WARNING - localhost:7000 - modeler - create_design - Closing previous design before_
↪creating a new one.

```

```

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...

```

Close the modeler

```

[8]: # Close the modeler
m.close()

```

```

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↪it was already running...

```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this exampleDownload this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.10 Modeling: Exporting designs

After creating a design, you typically want to bring it into a CAD tool for further development. This notebook demonstrates how to export a design to the various supported CAD formats.

Create a design

The code creates a simple design for demonstration purposes. The design consists of a set of rectangular pads with a circular hole in the center.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import UNITVECTOR3D_X, UNITVECTOR3D_Y, Point2D
from ansys.geometry.core.sketch import Sketch

# Instantiate the modeler
modeler = launch_modeler()

# Create a design
design = modeler.create_design("ExportDesignExample")

# Create a sketch
sketch = Sketch()

# Create a simple rectangle
sketch.box(Point2D([0, 0]), 10, 5)

# Extrude the sketch and displace the resulting body
# to make a plane of rectangles
for x_step in [-60, -45, -30, -15, 0, 15, 30, 45, 60]:
    for y_step in [-40, -30, -20, -10, 0, 10, 20, 30, 40]:
        # Extrude the sketch
        body = design.extrude_sketch(f"Body_X_{x_step}_Y_{y_step}", sketch, 5)

        # Displace the body in the x and y directions
        body.translate(UNITVECTOR3D_X, x_step)
        body.translate(UNITVECTOR3D_Y, y_step)

# Plot the design
design.plot()
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
port 700...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
a secure connection.")
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Export the design

You can export the design to various CAD formats. For the formats supported see the [DesignFileFormat](#) class, which is part of the the design module documentation.

Nonetheless, there are a set of convenience methods that you can use to export the design to the supported formats. The following code snippets demonstrate how to do it. You can decide whether to export the design to a file in a certain directory or in the current working directory.

Export to a file in the current working directory

```
[2]: # Export the design to a file in the current working directory
file_location = design.export_to_scdocx()
print(f"Design exported to {file_location}")

Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳examples/03_modeling/ExportDesignExample.scdocx
```

Export to a file in a certain directory

```
[3]: from pathlib import Path

# Define a downloads directory
download_dir = Path.cwd() / "downloads"

# Export the design to a file in a certain directory
file_location = design.export_to_scdocx(download_dir)
print(f"Design exported to {file_location}")

Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳examples/03_modeling/downloads/ExportDesignExample.scdocx
```

Export to SCDOCX format

```
[4]: # Export the design to a file in the requested directory
file_location = design.export_to_scdocx(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")

Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳examples/03_modeling/downloads/ExportDesignExample.scdocx
Does the file exist? True
```

Export to Parasolid text format

```
[5]: # Export the design to a file in the requested directory
file_location = design.export_to_parasolid_text(download_dir)
```

(continues on next page)

(continued from previous page)

```
# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

```
Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳ examples/03_modeling/downloads/ExportDesignExample.xmt_txt
Does the file exist? True
```

Export to Parasolid binary format

```
[6]: # Export the design to a file in the requested directory
file_location = design.export_to_parasolid_bin(download_dir)
```

```
# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

```
Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳ examples/03_modeling/downloads/ExportDesignExample.xmt_bin
Does the file exist? True
```

Export to STEP format

```
# Export the design to a file in the requested directory
file_location = design.export_to_step(download_dir)
```

```
# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Export to IGES format

```
# Export the design to a file in the requested directory
file_location = design.export_to_iges(download_dir)
```

```
# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

Export to FMD format

```
[7]: # Export the design to a file in the requested directory
file_location = design.export_to_fmd(download_dir)
```

```
# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

```
Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳ examples/03_modeling/downloads/ExportDesignExample.fmd
Does the file exist? True
```

Export to PMDB format

```
# Export the design to a file in the requested directory
file_location = design.export_to_pmdb(download_dir)
```

Export to Discovery format

```
[8]: # Export the design to a file in the requested directory
file_location = design.export_to_disco(download_dir)

# Print the file location
print(f"Design exported to {file_location}")
print(f"Does the file exist? {Path(file_location).exists()}")
```

```
Design exported to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
↳ examples/03_modeling/downloads/ExportDesignExample.dsco
Does the file exist? True
```

Close the modeler

Close the modeler after exporting the design.

```
[9]: # Close the modeler
modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↳ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.11 Modeling: Visualization of the design tree on terminal

A user can visualize its model object tree easily by using the `tree_print()` method available on the `Design` and `Component` objects. This method prints the tree structure of the model in the terminal.

Perform required imports

For the following example, you need to import these modules:

```
[1]: from pint import Quantity

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math.constants import UNITVECTOR3D_X, UNITVECTOR3D_Y
from ansys.geometry.core.math.point import Point2D, Point3D
from ansys.geometry.core.misc.units import UNITS
from ansys.geometry.core.sketch.sketch import Sketch
```

Create a design

The following code creates a simple design for demonstration purposes. The design consists of several cylinders extruded. The interesting part is visualizing the corresponding design tree.

```
[2]: # Create a modeler object
modeler = launch_modeler()

# Create your design on the server side
design = modeler.create_design("TreePrintComponent")

# Create a Sketch object and draw a circle (all client side)
sketch = Sketch()
sketch.circle(Point2D([-30, -30]), 10 * UNITS.m)
distance = 30 * UNITS.m

# The following component hierarchy is made
#
#           |---> comp_1 ---|---> nested_1_comp_1 ---> nested_1_nested_1_comp_1
#           |               |
#           |               |---> nested_2_comp_1
#           |
# DESIGN ---|---> comp_2 -----> nested_1_comp_2
#           |
#           |
#           |---> comp_3
#
#
# Now, only "comp_3", "nested_2_comp_1" and "nested_1_nested_1_comp_1"
# has a body associated.
#

# Create the components
comp_1 = design.add_component("Component_1")
comp_2 = design.add_component("Component_2")
comp_3 = design.add_component("Component_3")
nested_1_comp_1 = comp_1.add_component("Nested_1_Component_1")
nested_1_nested_1_comp_1 = nested_1_comp_1.add_component("Nested_1_Nested_1_Component_1")
nested_2_comp_1 = comp_1.add_component("Nested_2_Component_1")
nested_1_comp_2 = comp_2.add_component("Nested_1_Component_2")

# Create the bodies
```

(continues on next page)

(continued from previous page)

```

b1 = comp_3.extrude_sketch(name="comp_3_circle", sketch=sketch, distance=distance)
b2 = nested_2_comp_1.extrude_sketch(
    name="nested_2_comp_1_circle", sketch=sketch, distance=distance
)
b2.translate(UNITVECTOR3D_X, 50)
b3 = nested_1_nested_1_comp_1.extrude_sketch(
    name="nested_1_nested_1_comp_1_circle", sketch=sketch, distance=distance
)
b3.translate(UNITVECTOR3D_Y, 50)

```

```

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳ port 700...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳ on localhost:700. This is INSECURE. Consider using a secure connection.
    warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳ a secure connection.")

```

```

# Create beams (in design)
circle_profile_1 = design.add_beam_circular_profile(
    "CircleProfile1", Quantity(10, UNITS.mm), Point3D([0, 0, 0]), UNITVECTOR3D_X,
↳ UNITVECTOR3D_Y
)
beam_1 = nested_1_comp_2.create_beam(
    Point3D([9, 99, 999], UNITS.mm), Point3D([8, 88, 888], UNITS.mm), circle_profile_1
)

```

[3]: # Visualize the design (optional)

```

design.plot()

```

```

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv=&quot;Content-...

```

Visualize the design tree

Now, let's visualize the design tree using the `tree_print()` method. Let's start by printing the tree structure of the design object with no extra arguments.

[4]: `design.tree_print()`

```

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
:   |---(comp) Nested_1_Component_1
:   :   |---(comp) Nested_1_Nested_1_Component_1
:   :   :   |---(body) nested_1_nested_1_comp_1_circle

```

(continues on next page)

(continued from previous page)

```

: |---(comp) Nested_2_Component_1
: |---(body) nested_2_comp_1_circle
|---(comp) Component_2
: |---(comp) Nested_1_Component_2
|---(comp) Component_3
|---(body) comp_3_circle

```

Controlling the depth of the tree

The `tree_print()` method accepts an optional argument `depth` to control the depth of the tree to be printed. The default value is `None`, which means the entire tree is printed.

```
[5]: design.tree_print(depth=1)
```

```

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3

```

In this case, only the first level of the tree is printed - that is, the three main components.

Excluding bodies, components, or beams

By default, the `tree_print()` method prints all the bodies, components, and beams in the design tree. However, you can exclude any of these by setting the corresponding argument to `False`.

```
[6]: design.tree_print(consider_bodies=False, consider_beams=False)
```

```

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
: |---(comp) Nested_1_Component_1
: : |---(comp) Nested_1_Nested_1_Component_1
: |---(comp) Nested_2_Component_1
|---(comp) Component_2
: |---(comp) Nested_1_Component_2
|---(comp) Component_3

```

In this case, the bodies and beams are not be printed in the tree structure.

```
[7]: design.tree_print(consider_comps=False)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
```

In this case, the components are not be printed in the tree structure - leaving only the design object represented.

Sorting the tree

By default, the tree structure is sorted by the way the components, bodies, and beams were created. However, you can sort the tree structure by setting the `sort_keys` argument to `True`. In that case, the tree is sorted alphabetically.

Let's add a new component to the design and print the tree structure by default.

```
[8]: comp_4 = design.add_component("A_Component")
      design.tree_print(depth=1)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3
|---(comp) A_Component
```

Now, let's print the tree structure with the components sorted alphabetically.

```
[9]: design.tree_print(depth=1, sort_keys=True)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|---(comp) A_Component
|---(comp) Component_1
|---(comp) Component_2
|---(comp) Component_3
```

Indenting the tree

By default, the tree structure is printed with an indentation level of 4. However, you can indent the tree structure by setting the `indent` argument to the desired value.

```
[10]: design.tree_print(depth=1, indent=8)

>>> Tree print view of component 'TreePrintComponent'

Location
-----
Root component (Design)

Subtree
-----
(comp) TreePrintComponent
|----- (comp) Component_1
|----- (comp) Component_2
|----- (comp) Component_3
|----- (comp) A_Component
```

In this case, the tree structure is printed with an indentation level of 8.

Printing the tree from a specific component

You can print the tree structure from a specific component by calling the `tree_print()` method on the component object.

```
[11]: nested_1_comp_1.tree_print()

>>> Tree print view of component 'Nested_1_Component_1'

Location
-----
TreePrintComponent > Component_1 > Nested_1_Component_1

Subtree
-----
(comp) Nested_1_Component_1
|--- (comp) Nested_1_Nested_1_Component_1
|--- (body) nested_1_nested_1_comp_1_circle
```

Closing the modeler

Finally, close the modeler.

```
[12]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because...
↪ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.12 Modeling: Body color assignment and usage

In PyAnsys Geometry, a *body* represents solids or surfaces organized within the Design assembly. As users might be already familiar with, Ansys CAD products (like SpaceClaim, Ansys Discovery and the Geometry Service), allow to assign colors to bodies. This example shows how to assign colors to a body, retrieve their value and how to use them in the client-side visualization.

Perform required imports

Perform the required imports.

```
[1]: import ansys.geometry.core as pyansys_geometry

from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D, UNITVECTOR3D_X, UNITVECTOR3D_Y
from ansys.geometry.core.sketch import Sketch
```

Create a box sketch

Create a Sketch instance and insert a box sketch with a width and height of 10 in the default plane.

```
[2]: sketch = Sketch()
sketch.box(Point2D([0, 0]), 10, 10)

[2]: <ansys.geometry.core.sketch.sketch.Sketch at 0x7fcddbea8ad0>
```

Initiate design on server

Establish a server connection and initiate a design on the server.

```
[3]: modeler = launch_modeler()
design = modeler.create_design("ServiceColors")

WARNING - - docker_instance - _check_port_availability - Service is already running at_
↳ port 700...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS_
↳ on localhost:700. This is INSECURE. Consider using a secure connection.
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using_
↳ a secure connection.")
```

Extrude the box sketch to create the matrix style design

Given the initial sketch, you can extrude it to create a matrix style design. In this example, you can create a 2x3 matrix of bodies. Each body is separated by 30 units in the X direction and 30 units in the Y direction. You have a total of 6 bodies.

```
[4]: translate = [[0, 30, 60], [0, 30, 60]]

for r_idx, row in enumerate(translate):
    comp = design.add_component(f"Component{r_idx}")

    for b_idx, dist in enumerate(row):
        body = comp.extrude_sketch(f"Component{r_idx}_Body{b_idx}", sketch, distance=10)
        body.translate(UNITVECTOR3D_Y, r_idx*30)
        body.translate(UNITVECTOR3D_X, dist)

design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Assign colors to the bodies

Given the previous design, you can assign a color to each body. You could have done this assignment while creating the bodies, but for the sake of encapsulating the color assignment logic, it is done in its own code cell.

```
[5]: colors = ["red", "blue", "yellow"], ["orange", "green", "purple"]

for c_idx, comp in enumerate(design.components):
    for b_idx, body in enumerate(comp.bodies):
        body.color = colors[c_idx][b_idx]
        print(f"Body {body.name} has color {body.color}")

Body Component0_Body0 has color #ff0000ff
Body Component0_Body1 has color #0000ffff
Body Component0_Body2 has color #ffff00ff
Body Component1_Body0 has color #ffa500ff
Body Component1_Body1 has color #008000ff
Body Component1_Body2 has color #800080ff
```

Plotting the design with colors

By default, the plot method does **not** use the colors assigned to the bodies. To plot the design with the assigned colors, you need to specifically request it.

Users have two options for plotting with the assigned colors:

- Pass the parameter `use_service_colors=True` to the plot method.
- Set the global parameter `USE_SERVICE_COLORS` to `True`.

It is important to note that the usage of colors when plotting might slow down the plotting process, as it requires additional information to be sent from the server to the client and processed in the client side.

If you just request the plot without setting the global parameter, the plot will be displayed without the colors, as shown below.

```
[6]: design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

As stated previously, if you pass the parameter `use_service_colors=True` to the plot method, the plot is displayed with the assigned colors.

```
[7]: design.plot(use_service_colors=True)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

However, if you set the global parameter to `True`, the plot is displayed with the assigned colors without the need to pass the parameter to the plot method.

```
[8]: import ansys.geometry.core as pyansys_geometry
```

```
pyansys_geometry.USE_SERVICE_COLORS = True
```

```
design.plot()
```

```
# Reverting the global parameter to its default value
```

```
pyansys_geometry.USE_SERVICE_COLORS = False
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

This last method is useful when the user wants to plot all the designs with the assigned colors without the need to pass the parameter to the plot method in every call.

Plotting specific bodies or components with colors

If the user wants to plot specific bodies with the assigned colors, the user can follow the same approach as before. The user can pass the parameter `use_service_colors=True` to the plot method or set the global parameter `USE_SERVICE_COLORS` to `True`.

In the following examples, you are shown how to do this using the `use_service_colors=True` parameter.

Let's plot the first body of the first component with the assigned colors.

```
[9]: body = design.components[0].bodies[0]
```

```
body.plot(use_service_colors=True)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Now, let's plot the second component with the assigned colors.

```
[10]: comp = design.components[1]
```

```
comp.plot(use_service_colors=True)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.13 Modeling: Surface bodies and trimmed surfaces

This example shows how to trim different surfaces, and how to use those surfaces to create surface bodies.

Create a surface

Create a sphere surface. This can be done without launching the modeler.

```
[1]: from ansys.geometry.core.shapes-surfaces import Sphere
from ansys.geometry.core.math import Point3D

surface = Sphere(origin=Point3D([0, 0, 0]), radius=1)
```

Now get information on how the surface is defined and parameterized.

```
[2]: surface.parameterization()

[2]: (Parameterization(form=ParamForm.PERIODIC, type=ParamType.CIRCULAR,
↪ interval=Interval(start=0, end=6.283185307179586)),
Parameterization(form=ParamForm.CLOSED, type=ParamType.OTHER, interval=Interval(start=-
↪ 1.5707963267948966, end=1.5707963267948966)))
```

Trim the surface

For a sphere, its parametrization is (u: [0, 2*pi], v: [-pi/2, pi/2]), where u corresponds to longitude and v corresponds to latitude. You can **trim** a surface by providing new parameters.

```
[3]: from ansys.geometry.core.shapes-box_uv import BoxUV
from ansys.geometry.core.shapes-parameterization import Interval
import math

trimmed_surface = surface.trim(BoxUV(range_u=Interval(0, math.pi), range_v=Interval(0,
↪ math.pi/2)))
```

From a TrimmedSurface, you can always refer back to the underlying Surface if needed.

```
[4]: trimmed_surface.geometry

[4]: <ansys.geometry.core.shapes-surfaces.sphere.Sphere at 0x7f9d8c83fcb0>
```

Create a surface body

Now create a surface body by launching the modeler session and providing the trimmed surface. Then plot the body to see how you created a quarter of a sphere as a surface body.

```
[5]: from ansys.geometry.core import launch_modeler

modeler = launch_modeler()
print(modeler)
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 7000...
```

```
Ansys Geometry Modeler (0x7f9d8c266660)

Ansys Geometry Modeler Client (0x7f9d8c32aa50)
Target:      localhost:7000
Connection: Healthy
Backend info:
  Version:           27.1.0
  Backend type:      CORE_LINUX
  Backend number:    20260130.2
  API server number: 1984
  CADIntegration:    27.1.0.7
```

```
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")
```

```
[6]: design = modeler.create_design("SurfaceBodyExample")
body = design.create_body_from_surface("trimmed_sphere", trimmed_surface)
design.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳http-equiv=&quot;Content-...
```

If the sphere was left untrimmed, it would create a solid body since the surface is fully closed. In this case, since the surface was open, it created a surface body.

This same process can be used with other surfaces including:

- Cone
- Cylinder
- Plane
- Torus

Each surface has its own unique parameterization, which must be understood before trying to trim it.

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.


```
[7]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↳ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.14 Modeling: Using design parameters

You can read and update parameters that are part of the design. The simple design in this example has two associated parameters.

Perform required imports

```
[1]: from pathlib import Path
import requests

from ansys.geometry.core import launch_modeler
```

The file for this example is in the integration tests folder and can be downloaded.

Download the example file

Download the file for this example from the integration tests folder in the PyAnsys Geometry repository.

```
[2]: def download_file(url, filename):
    """Download a file from a URL and save it to a local file."""
    response = requests.get(url)
    response.raise_for_status() # Check if the request was successful
    with open(filename, 'wb') as file:
        file.write(response.content)

# URL of the file to download
url = "https://raw.githubusercontent.com/ansys/pyansys-geometry/main/tests/integration/
↳ files/blockswithparameters.dsco"

# Local path to save the file to
file_path = Path.cwd() / "blockswithparameters.dsco"

# Download the file
download_file(url, file_path)
print(f"File is downloaded to {file_path}")
```

File is downloaded to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/
 ↪examples/03_modeling/blockswithparameters.dsco

Import a design with parameters

Import the model using the `open_file()` method of the modeler.

```
[3]: # Create a modeler object
modeler = launch_modeler()
design = modeler.open_file(file_path)
design.plot()

WARNING - - docker_instance - _check_port_availability - Service is already running at
↪port 7000...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↪ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↪on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↪a secure connection.")

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪http-equiv=&quot;Content-...
```

Read existing parameters of the design

You can get all the parameters of the design as a list of parameters. Because this example has two parameters, you see two items in the list.

```
[4]: my_parameters = design.parameters
print(len(my_parameters))

4
```

A parameter object has a name, value, and unit.

```
[5]: print(my_parameters[0].name)
print(my_parameters[0].dimension_value)
print(my_parameters[0].dimension_type)

print(my_parameters[1].name)
print(my_parameters[1].dimension_value)
print(my_parameters[1].dimension_type)

p1
0.00010872999999999982 meter ** 2
ParameterType.DIMENSIONTYPE_AREA
p2
0.0002552758322160814 meter ** 2
ParameterType.DIMENSIONTYPE_AREA
```

Parameter values are returned in the default unit for each dimension type. Since default length unit is meter and default area unit is meter square, the value is returned in square meters.

Edit a parameter value

You can edit the parameter's name or value by simply setting these fields. Set the second parameter (p2 value to 350 mm).

```
[6]: parameter1 = my_parameters[1]
parameter1.dimension_value = 0.000440
response = design.set_parameter(parameter1)
print(response)
print(my_parameters[0].dimension_value)
print(my_parameters[1].dimension_value)
```

```
ParameterUpdateStatus.SUCCESS
0.00010872999999999982 meter ** 2
0.00044 meter ** 2
```

After a successful parameter update, the design is updated. If we request the design plot again, we see the updated design.

```
[7]: design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

The `set_parameter()` method returns a `Success` status message if the parameter is updated or a `FAILURE` status message if the update fails. If the p2 parameter depends on the p1 parameter, updating the p1 parameter might also change the p2 parameter. In such cases, the method returns `CONSTRAINED_PARAMETERS`, which indicates other parameters were also updated.

```
[8]: parameter1 = my_parameters[0]
parameter1.dimension_value = 0.000250
response = design.set_parameter(parameter1)
print(response)
```

```
ParameterUpdateStatus.CONSTRAINED_PARAMETERS
```

To get the updated list, query the parameters once again.

```
[9]: my_parameters = design.parameters
print(my_parameters[0].dimension_value)
print(my_parameters[1].dimension_value)
```

```
0.0002499999999999997263 meter ** 2
0.000581269999999999444 meter ** 2
```

Close the modeler

Close the modeler to free up resources and release the connection.

```
[10]: modeler.close()

WARNING - localhost:700 - client - close - Geometry service was not shut down because_
↪it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.15 Modeling: Chamfer edges and faces

A chamfer is an angled cut on an edge. Chamfers can be created using the `Modeler.geometry_commands` module.

Create a block

Launch the modeler and create a block.

```
[1]: from ansys.geometry.core import launch_modeler, Modeler

modeler = launch_modeler()
print(modeler)

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 700...

Ansys Geometry Modeler (0x7f19b2730440)

Ansys Geometry Modeler Client (0x7f19b2679400)
Target:      localhost:700
Connection: Healthy
Backend info:
  Version:           27.1.0
  Backend type:      CORE_LINUX
  Backend number:    20260130.2
  API server number: 1984
  CADIntegration:    27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")

[2]: from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import Point2D

design = modeler.create_design("chamfer_block")
body = design.extrude_sketch("block", Sketch().box(Point2D([0, 0]), 1, 1), 1)

body.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Chamfer edges

Create a uniform chamfer on all edges of the block.

```
[3]: modeler.geometry_commands.chamfer(body.edges, distance=0.1)
```

```
body.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Chamfer faces

The chamfer of a face can also be modified. Create a chamfer on a single edge and then modify the chamfer distance value by providing the newly created face that represents the chamfer.

```
[4]: body = design.extrude_sketch("box", Sketch().box(Point2D([0,0]), 1, 1), 1)
```

```
modeler.geometry_commands.chamfer(body.edges[0], distance=0.1)
```

```
body.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

```
[5]: modeler.geometry_commands.chamfer(body.faces[-1], distance=0.3)
```

```
body.plot()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

Close the server session.

```
[6]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↪it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.3.16 Modeling: Detach faces

This example shows how to use the detach faces operations to separate faces from bodies and convert them into independent surface bodies.

The detach operation is useful when you need to:

- Extract specific faces from a solid body for further manipulation
- Create surface bodies from existing geometry
- Break down complex geometries into simpler components

Perform required imports

Perform the required imports.

```
[1]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Plane, Point2D, Point3D
from ansys.geometry.core.misc import UNITS
from ansys.geometry.core.sketch import Sketch
```

Launch modeler

Launch the modeler.

```
[2]: modeler = launch_modeler()
print(modeler)
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at
↳port 700...
```

```
Ansys Geometry Modeler (0x7f6880389160)

Ansys Geometry Modeler Client (0x7f68802c5400)
Target:      localhost:700
Connection: Healthy
Backend info:
  Version:      27.1.0
  Backend type: CORE_LINUX
  Backend number: 20260130.2
  API server number: 1984
  CADIntegration: 27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳a secure connection.")
```

Detach faces using geometry commands

The `modeler.geometry_commands.detach_faces()` method allows you to detach faces from one or more bodies or faces. This creates new surface bodies for each detached face.

Detach a single face

Create a box and detach a single face from it.

```
[3]: # Create a design and a box
design = modeler.create_design("detach_single_face")
box = design.extrude_sketch("box", Sketch().box(Point2D([0, 0]), 1, 1), 1)

print(f"Initial number of bodies: {len(design.bodies)}")
print(f"Number of faces on box: {len(box.faces)}")

# Plot the original box
box.plot()

Initial number of bodies: 1
Number of faces on box: 6

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

```
[4]: # Select the top face (typically the last face)
top_face = box.faces[-1]

# Detach the single face
created_bodies = modeler.geometry_commands.detach_faces(top_face)

# Update the color of detached bodies for better visibility in the plot.
for body in created_bodies:
    body.color = "red"

print(f"Number of bodies created: {len(created_bodies)}")
print(f"Total number of bodies: {len(design.bodies)}")

# Plot the design to see the detached face
design.plot(use_service_colors=True)

Number of bodies created: 1
Total number of bodies: 2

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

The detached face is now a separate surface body that can be manipulated independently.

Detach multiple selected faces

You can also detach multiple faces at once by passing a list of faces.

```
[5]: # Create a new box
design = modeler.create_design("detach_multiple_faces")
box = design.extrude_sketch("box", Sketch().box(Point2D([0, 0]), 2, 2), 2)
```

(continues on next page)

(continued from previous page)

```

print(f"Initial number of bodies: {len(design.bodies)}")

# Select three faces to detach
faces_to_detach = [box.faces[0], box.faces[1], box.faces[2]]

# Detach the selected faces
created_bodies = modeler.geometry_commands.detach_faces(faces_to_detach)

# Update the color of detached bodies for better visibility in the plot.
for body in created_bodies:
    body.color = "red"

print(f"Number of surface bodies created: {len(created_bodies)}")
print(f"Total number of bodies: {len(design.bodies)}")

# Verify that all created bodies are surface bodies
for i, body in enumerate(created_bodies):
    print(f"Body {i}: is_surface = {body.is_surface}, faces = {len(body.faces)}")

# Plot the design
design.plot(use_service_colors=True)

```

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳ creating a new one.

Initial number of bodies: 1
Number of surface bodies created: 1
Total number of bodies: 2
Body 0: is_surface = True, faces = 3

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n <head>\n <meta_

↳ http-equiv="Content-...

Detach from multiple bodies

The `detach_faces` method can also accept a list of bodies, detaching all faces from each body.

```

[6]: # Create a design with two boxes
design = modeler.create_design("detach_from_multiple_bodies")
box1 = design.extrude_sketch("box1", Sketch().box(Point2D([0, 0]), 1, 1), 1)
box2 = design.extrude_sketch("box2", Sketch().box(Point2D([2, 0]), 1, 1), 1)

print(f"Initial number of bodies: {len(design.bodies)}")

# Detach all faces from both boxes
created_bodies = modeler.geometry_commands.detach_faces([box1, box2])

# Update the color of detached bodies for better visibility in the plot.
for body in created_bodies:
    body.color = "red"

print(f"Number of surface bodies created: {len(created_bodies)}")

```

(continues on next page)

(continued from previous page)

```
print(f"Total number of bodies: {len(design.bodies)}")

# Plot the design
design.plot(use_service_colors=True)

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳ creating a new one.

Initial number of bodies: 2
Number of surface bodies created: 10
Total number of bodies: 12

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv="Content-...
```

Each box has 6 faces, so detaching from two boxes creates 12 new surface bodies.

Detach faces using the Body method

You can call the `detach_faces()` method directly on a `Body` object. This is equivalent to calling `modeler.geometry_commands.detach_faces(body)`.

```
[7]: # Create a new design with a box
design = modeler.create_design("body_detach_faces")
box = design.extrude_sketch("box", Sketch().box(Point2D([0, 0]), 1.5, 1.5), 1.5)

print(f"Initial number of bodies: {len(design.bodies)}")
print(f"Box volume: {box.volume}")

# Plot the original box
box.plot()

WARNING - localhost:7000 - modeler - create_design - Closing previous design before
↳ creating a new one.

Initial number of bodies: 1
Box volume: 3.375 meter ** 3

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↳ http-equiv="Content-...
```

```
[8]: # Call detach_faces directly on the body
created_bodies = box.detach_faces()

# Update the color of detached bodies for better visibility in the plot.
for body in created_bodies:
    body.color = "red"

print(f"Number of surface bodies created: {len(created_bodies)}")
print(f"Total number of bodies: {len(design.bodies)}")

# Check that all created bodies are surface bodies
for body in created_bodies:
    print(f"Body '{body.name}': is_surface = {body.is_surface}")
```

(continues on next page)

(continued from previous page)

```
# Plot the design with all detached faces
design.plot(use_service_colors=True)

Number of surface bodies created: 5
Total number of bodies: 6
Body 'Surface': is_surface = True
Body 'Surface': is_surface = True
Body 'Surface': is_surface = True
Body 'Surface': is_surface = True
Body 'Surface': is_surface = True

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Detach a single face using the Face method

Individual faces can also be detached using the `detach()` method directly on a Face object.

```
[9]: # Create a cylinder for variety
design = modeler.create_design("face_detach")
sketch = Sketch()
sketch.circle(Point2D([0, 0]), 0.5)
cylinder = design.extrude_sketch("cylinder", sketch, 2)

print(f"Initial number of bodies: {len(design.bodies)}")
print(f"Number of faces on cylinder: {len(cylinder.faces)}")

# Plot the original cylinder
cylinder.plot()

WARNING - localhost:7000 - modeler - create_design - Closing previous design before_
↳creating a new one.

Initial number of bodies: 1
Number of faces on cylinder: 3

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

A cylinder has 3 faces: top, bottom, and the cylindrical surface.

```
[10]: # Select the top face (circular face)
top_face = cylinder.faces[0]
print(f"Top face area: {top_face.area}")

# Detach the face
result = top_face.detach()

if result is not None:
    print(f"face detached successfully.")
print(f"Total number of bodies: {len(design.bodies)}")

# Update the color of detached body for better visibility in the plot.
result.color = "red"
```

(continues on next page)

(continued from previous page)

```
# The result is a list containing the created surface body
if result is not None:
    print(f"Created body is surface: {result.is_surface}")
    print(f"Created body face area: {result.faces[0].area}")

# Plot the design
design.plot(use_service_colors=True)
```

```
Top face area: 0.7853981633974482 meter ** 2
face detached successfully.
Total number of bodies: 2
Created body is surface: True
Created body face area: 0.7853981633974482 meter ** 2
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Practical use case: Extract specific faces for analysis

This example demonstrates a practical use case where you extract specific faces from a complex geometry for further analysis or manipulation.

```
[11]: # Create a more complex geometry
design = modeler.create_design("practical_example")

# Create a base box
base = design.extrude_sketch("base", Sketch().box(Point2D([0, 0]), 3, 3), 0.5)

# Create a smaller box on top
top_sketch = Sketch(Plane(origin=Point3D([0, 0, 0.5])))
top_sketch.box(Point2D([0.5, 0.5]), 2, 2)
top = design.extrude_sketch("top", top_sketch, 1)

print(f"Initial number of bodies: {len(design.bodies)}")

# Plot the original geometry
design.plot(use_service_colors=True)
```

```
WARNING - localhost:7000 - modeler - create_design - Closing previous design before_
↳creating a new one.
```

```
Initial number of bodies: 2
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

```
[12]: # Unite the two boxes
base.unite(top)

print(f"After union - number of bodies: {len(design.bodies)}")
print(f"Number of faces on combined body: {len(base.faces)}")

# Plot the combined geometry
base.plot()
```

```
After union - number of bodies: 1
Number of faces on combined body: 9
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

```
[13]: # Extract the top face for analysis
# In this case, we'll look for the face with the highest z-coordinate
top_faces = [face for face in base.faces]

# Detach the top planar face
if top_faces:
    top_face = top_faces[-1] # Usually the last one
    extracted_surface = top_face.detach()
    extracted_surface.color = "red"
    print(f"Extracted surface body: {extracted_surface.name}")
    print(f"Is surface: {extracted_surface.is_surface}")
    print(f"Total bodies in design: {len(design.bodies)}")

# Plot the final design with the extracted surface
design.plot(use_service_colors=True)
```

```
Extracted surface body: Surface
Is surface: True
Total bodies in design: 2
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Close session

When you finish interacting with your modeling service, you should close the active server session. This frees resources wherever the service is running.

```
[14]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↪it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.4 Applied examples

These examples demonstrate the usage of PyAnsys Geometry for real-world applications.

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.4.1 Applied: Create a NACA 4-digit airfoil

NACA airfoils are a series of airfoil shapes for aircraft wings developed by the National Advisory Committee for Aeronautics (NACA). They are a standardized system of airfoil shapes that are defined by a series of digits. The digits, which indicate the shape of the airfoil, are used to create the airfoil shape.

Each digit in the NACA airfoil number has a specific meaning:

- The first digit defines the maximum camber as a percentage of the chord length.
- The second digit defines the position of the maximum camber as a percentage of the chord length.
- The last two digits define the maximum thickness of the airfoil as a percentage of the chord length.

To fully understand the previous definitions, it is important to know that the chord length is the length of the airfoil from the leading edge to the trailing edge. The camber is the curvature of the airfoil, and the thickness is the distance between the upper and lower surfaces.

Symmetric airfoils have a camber of 0% and consequently, the first two digits of the NACA number are 0. For example, the NACA 0012 airfoil is a symmetric airfoil with a maximum thickness of 12%.

Define the NACA 4-digit airfoil equation

The following code uses the equation for a NACA 4-digit airfoil to create a set of points that define the airfoil shape. These points are `Point2D` objects in PyAnsys Geometry.

```
[1]: from typing import List, Union

import numpy as np

from ansys.geometry.core.math import Point2D

def naca_airfoil_4digits(number: Union[int, str], n_points: int = 200) -> list[Point2D]:
    """
    Generate a NACA 4-digits airfoil.

    Parameters
    -----
    number : int or str
        NACA 4-digit number.
    n_points : int
        Number of points to generate the airfoil. The default is ``200``.
        Number of points in the upper side of the airfoil.
        The total number of points is ``2 * n_points - 1``.

    Returns
    -----
    list[Point2D]
        List of points that define the airfoil.
    """
    # 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
```

(continues on next page)

(continued from previous page)

```

t = number % 100 * 0.01

# Generate the airfoil
points = []
for i in range(n_points):

    # Make it a exponential distribution so the points are more concentrated
    # near the leading edge
    x = (1 - np.cos(i / (n_points - 1) * np.pi)) / 2

    # Check if it is a symmetric airfoil
    if p == 0 and m == 0:
        # Camber line is zero in this case
        yc = 0
        dyc_dx = 0
    else:
        # Compute the camber line
        if x < p:
            yc = m / p**2 * (2 * p * x - x**2)
            dyc_dx = 2 * m / p**2 * (p - x)
        else:
            yc = m / (1 - p) ** 2 * ((1 - 2 * p) + 2 * p * x - x**2)
            dyc_dx = 2 * m / (1 - p) ** 2 * (p - x)

    # Compute the thickness
    yt = 5 * t * (0.2969 * x**0.5
                  - 0.1260 * x
                  - 0.3516 * x**2
                  + 0.2843 * x**3
                  - 0.1015 * x**4)

    # Compute the angle
    theta = np.arctan(dyc_dx)

    # Compute the points (upper and lower side of the airfoil)
    xu = x - yt * np.sin(theta)
    yu = yc + yt * np.cos(theta)
    xl = x + yt * np.sin(theta)
    yl = yc - yt * np.cos(theta)

    # Append the points
    points.append(Point2D([xu, yu]))
    points.insert(0, Point2D([xl, yl]))

    # Remove the first point since it is repeated
    if i == 0:
        points.pop(0)

return points

```

Example of a symmetric airfoil: NACA 0012

Now that the function for generating a NACA 4-digit airfoil is generated, this code creates a NACA 0012 airfoil, which is symmetric. This airfoil has a maximum thickness of 12%. The NACA number is a constant.

```
[2]: NACA_AIRFOIL = "0012"
```

Required imports

Before you start creating the airfoil points, you must import the necessary modules to create the airfoil using PyAnsys Geometry.

```
[3]: from ansys.geometry.core import launch_modeler
     from ansys.geometry.core.sketch import Sketch
```

Generate the airfoil points

Using the function defined previously, you generate the points that define the NACA 0012 airfoil. Create a Sketch object and add the points to it. Then, approximate the airfoil using straight lines between the points.

```
[4]: # Create a sketch
     sketch = Sketch()

     # Generate the points of the airfoil
     points = naca_airfoil_4digits(NACA_AIRFOIL)

     # Create the segments of the airfoil
     for i in range(len(points) - 1):
         sketch.segment(points[i], points[i + 1])

     # Close the airfoil
     sketch.segment(points[-1], points[0])

     # Plot the airfoil
     sketch.plot()

     EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
     ↪http-equiv=&quot;Content-...
```

Create the 3D airfoil

Once the Sketch object is created, you create a 3D airfoil. For this operation, you must create a modeler object, create a design object, and extrude the sketch.

```
[5]: # Launch the modeler
     modeler = launch_modeler()

     # Create the design
     design = modeler.create_design(f"NACA_Airfoil_{NACA_AIRFOIL}")

     # Extrude the airfoil
     design.extrude_sketch("Airfoil", sketch, 1)
```

(continues on next page)

(continued from previous page)

```
# Plot the design
design.plot()

WARNING - - docker_instance - _check_port_availability - Service is already running at
↳ port 7000...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS
↳ on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↳ a secure connection.")

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 /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/examples/
↳ 04_applied/NACA_Airfoil_0012.fmd
```

Close the modeler

```
[7]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↳ it was already running...
```

Example of a cambered airfoil: NACA 6412

This code creates a NACA 6412 airfoil, which is cambered. This airfoil has a maximum camber of 6% and a maximum thickness of 12%. After overriding the NACA number, the code generates the airfoil points.

```
[8]: NACA_AIRFOIL = "6412"
```

Generate the airfoil points

As before, you generate the points that define the NACA 6412 airfoil. Create a `Sketch` object and add the points to it. Then, approximate the airfoil using straight lines.

```
[9]: # Create a sketch
sketch = Sketch()

# Generate the points of the airfoil
points = naca_airfoil_4digits(NACA_AIRFOIL)

# Create the segments of the airfoil
for i in range(len(points) - 1):
```

(continues on next page)

(continued from previous page)

```

        sketch.segment(points[i], points[i + 1])

# Close the airfoil
sketch.segment(points[-1], points[0])

# Plot the airfoil
sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Create the 3D airfoil

```

[10]: # Launch the modeler
modeler = launch_modeler()

# Create the design
design = modeler.create_design(f"NACA_Airfoil_{NACA_AIRFOIL}")

# Extrude the airfoil
design.extrude_sketch("Airfoil", sketch, 1)

# Plot the design
design.plot()

WARNING - - docker_instance - _check_port_availability - Service is already running at_
↳port 7000...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS_
↳on localhost:7000. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using_
↳a secure connection.")

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Save the design

In this case, the design is saved as an SCDOCX file.

```

[11]: # Save the design
file = design.export_to_scdocx()
print(f"Design saved to {file}")

Design saved to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/examples/
↳04_applied/NACA_Airfoil_6412.scdocx
```

Close the modeler

```

[12]: modeler.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↳it was already running...
```

(continues on next page)

(continued from previous page)

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because...
↳ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.4.2 Applied: Prepare a NACA airfoil for a Fluent simulation

Once a NACA airfoil is designed, it is necessary to prepare the geometry for a CFD simulation. This notebook demonstrates how to prepare a NACA 6412 airfoil for a Fluent simulation. The starting point of this example is the previously designed NACA 6412 airfoil. The airfoil was saved in an SCDOCX file, which is now imported into the notebook. The geometry is then prepared for the simulation.

In case you want to run this notebook, make sure that you have run the previous notebook to design the NACA 6412 airfoil.

Import the NACA 6412 airfoil

The following code starts up the Geometry Service and imports the NACA 6412 airfoil. The airfoil is then displayed in the notebook.

```
[1]: import os

from ansys.geometry.core import launch_modeler

# Launch the modeler
modeler = launch_modeler()

# Import the NACA 6412 airfoil
design = modeler.open_file(os.path.join(os.getcwd(), f"NACA_Airfoil_6412.scdocx"))

# Retrieve the airfoil body
airfoil = design.bodies[0]

# Display the airfoil
design.plot()
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at...
↳ port 7000...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↳ ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS...
↳ on localhost:7000. This is INSECURE. Consider using a secure connection.
```

(continues on next page)

(continued from previous page)

```
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↪ a secure connection.")

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪ http-equiv=&quot;Content-
```

Prepare the geometry for the simulation

The current design is only composed of the airfoil. To prepare the geometry for the simulation, you must define the domain around the airfoil. The following code creates a rectangular fluid domain around the airfoil.

The airfoil has the following dimensions:

- Chord length: 1 (X-axis)
- Thickness: Depends on NACA value (Y-axis)

Define the fluid domain as a large box with these dimensions:

- Length (X-axis) - 10 times the chord length
- Width (Z-axis) - 5 times the chord length
- Height (Y-axis) - 4 times the chord length

Place the airfoil at the center of the fluid domain.

```
[2]: from ansys.geometry.core.math import Point2D, Plane, Point3D
from ansys.geometry.core.sketch import Sketch

BOX_LENGTH = 10 # X-Axis
BOX_WIDTH = 5 # Z-Axis
BOX_HEIGHT = 4 # Y-Axis

# Create the sketch
fluid_sketch = Sketch(
    plane=Plane(origin=Point3D([0, 0, 0.5 - (BOX_WIDTH / 2)]))
)
fluid_sketch.box(
    center=Point2D([0.5, 0]),
    height=BOX_HEIGHT,
    width=BOX_LENGTH,
)

# Extrude the fluid domain
fluid = design.extrude_sketch("Fluid", fluid_sketch, BOX_WIDTH)
```

Create named selections

Named selections are used to define boundary conditions in Fluent. The following code creates named selections for the inlet, outlet, and walls of the fluid domain. The airfoil is also assigned a named selection.

The airfoil is aligned with the X axis. The inlet is located at the left side of the airfoil, the outlet is located at the right side of the airfoil, and the walls are located at the top and bottom of the airfoil. The inlet face has therefore a negative X-axis normal vector, and the outlet face has a positive X-axis normal vector. The rest of the faces, therefore, constitute the walls.

```
[3]: # Create named selections in the fluid domain (inlet, outlet, and surrounding faces)
# Add also the airfoil as a named selection
fluid_faces = fluid.faces
surrounding_faces = []
inlet_faces = []
outlet_faces = []
for face in fluid_faces:
    if face.normal().x == 1:
        outlet_faces.append(face)
    elif face.normal().x == -1:
        inlet_faces.append(face)
    else:
        surrounding_faces.append(face)

design.create_named_selection("Outlet Fluid", faces=outlet_faces)
design.create_named_selection("Inlet Fluid", faces=inlet_faces)
design.create_named_selection("Surrounding Faces", faces=surrounding_faces)
design.create_named_selection("Airfoil Faces", faces=airfoil.faces)
```

```
[3]: ansys.geometry.core.designer.selection.NamedSelection 0x7fb3d87bd5b0
Name          : Airfoil Faces
Id            : 1:72
N Bodies      : 0
N Faces       : 399
N Edges       : 0
N Beams       : 0
N Design Points : 0
N Components  : 0
N Vertices    : 0
```

Display the geometry

The geometry is now ready for the simulation. The following code displays the geometry in the notebook. This example uses the `GeometryPlotter` class to display the geometry for the airfoil and fluid domain in different colors with a specified opacity level.

The airfoil is displayed in green, and the fluid domain is displayed in blue with an opacity of 0.25.

```
[4]: from ansys.geometry.core.plotting import GeometryPlotter

plotter = GeometryPlotter()
plotter.plot(airfoil, color="green")
plotter.plot(fluid, color="blue", opacity=0.15)
plotter.show()
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↪http-equiv=&quot;Content-...
```

Export the geometry

Export the geometry into a Fluent-compatible format. The following code exports the geometry into a PMDB file, which retains the named selections.

```
# Save the design
file = design.export_to_pmdb()
```

You can import the exported PMDB file into Fluent to set up the mesh and perform the simulation. For an example of how to set up the mesh and boundary conditions in Fluent, see the [Modeling External Compressible Flow](#) example in the Fluent documentation.

The main difference between the Fluent example and this geometry is the coordinate system. The Fluent example defines the airfoil in the XY plane, while this geometry defines the airfoil in the XZ plane.

Close the modeler

```
[5]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because...  
↪ it was already running...
```

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.4.3 Applied: Develop an external aerodynamic simulation model for Fluent analysis

The Ahmed body is a simplified car model used to study airflow around vehicles. The wake (turbulent flow behind the body) depends on the slant angle:

- Less than 12 degrees: The airflow stays attached to the slant, creating low drag and a mostly two-dimensional flow.
- 12 to 30 degrees: The flow becomes three-dimensional with strong c-pillar vortices, peaking at 30 degrees. This increases drag due to low-pressure areas on the rear surfaces.
- More than 30 degrees: The flow fully separates from the slant, reducing drag and weakening the c-pillar vortices.

This example creates an Ahmed body with a slant angle of 20 degrees. It consists of these steps:

1. Launch PyAnsys Geometry and define the default units.
2. Create sketches for the Ahmed body, enclosure, and BOI (Body of Influence).
3. Generate solid bodies from the sketches.
4. Perform Boolean operations for region extraction.
5. Group faces and define a named selection.
6. Export model as a CAD file.
7. Close session.

Define function for sorting planar face pairs along any axis

This function is used to sort the planar faces along any of the coordinate axis. This is used primarily for sorting the faces to define the named selections.

```
[1]: def face_identifier(faces, axis):
    """
    Sort a pair of planar faces based on their positions along the specified coordinate_
    ↪axis.

    Args:
        faces : List[IFace, IFace]
        List of planar face pairs.

        axis : (string)
        Axis to sort the face pair on. Options are "x", "y", or "z".

    Returns:
        IFace, IFace
        - IFace: Face with the centroid positioned behind the other face along the_
    ↪specified axis.
        - IFace: Face with the centroid positioned ahead of the other face along the_
    ↪specified axis.

    """
    min_face = ""
    max_face = ""
    if axis == "x":
        position = 0
    elif axis == "y":
        position = 1
    else:
        position = 2
    min_face_cor_val = faces[0].point(0.5, 0.5)[position]
    min_face = faces[0]
    max_face_cor_val = faces[0].point(0.5, 0.5)[position]
    max_face = faces[0]
    for face in faces[1:]:
        if face.point(0.5, 0.5)[position] < min_face_cor_val:
            min_face_cor_val = face.point(0.5, 0.5)[position]
            min_face = face
            continue
        elif face.point(0.5, 0.5)[position] > max_face_cor_val:
            max_face_cor_val = face.point(0.5, 0.5)[position]
            max_face = face
    return min_face, max_face
```

Define function for calculating the vertical and horizontal distances based on the slant angle

```
[2]: def distance_calculator(hypo, slant_angle):
    """
    Calculate the horizontal and vertical distances based on the hypotenuse and slant_
```

(continues on next page)

(continued from previous page)

```

↪angle.

    Args:
        hypo : int
            Length of the hypotenuse in millimeters.

        slant_angle : int
            Slant angle in degrees.

    Returns:
        slant_x (float): Horizontal distance calculated using the sine of the slant_
↪angle.
        slant_y (float): Vertical distance calculated using the cosine of the slant_
↪angle.

    """
    slant_x = hypo * math.cos(math.radians(slant_angle))
    slant_y = hypo * math.sin(math.radians(slant_angle))
    return slant_y, slant_x

```

Launch PyAnsys geometry and define the default units

Before you start creating the Ahmed body, you must import the necessary modules to create the model using PyAnsys Geometry. It's also a good practice to define the units before initiating the development of the sketch.

```

[3]: from ansys.geometry.core import launch_modeler
from ansys.geometry.core.sketch import Sketch
from ansys.geometry.core.math import (
    Point2D,
    Plane,
    Point3D,
    UNITVECTOR3D_X,
    UNITVECTOR3D_Y,
    UNITVECTOR3D_Z,
)
from ansys.geometry.core.misc import UNITS, DEFAULT_UNITS

from ansys.geometry.core.plotting import GeometryPlotter
import math
import os

modeler = launch_modeler()
DEFAULT_UNITS.LENGTH = UNITS.mm
DEFAULT_UNITS.angle = UNITS.degrees

```

```

WARNING - - docker_instance - _check_port_availability - Service is already running at_
↪port 700...
/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↪ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS_
↪on localhost:700. This is INSECURE. Consider using a secure connection.

```

(continues on next page)

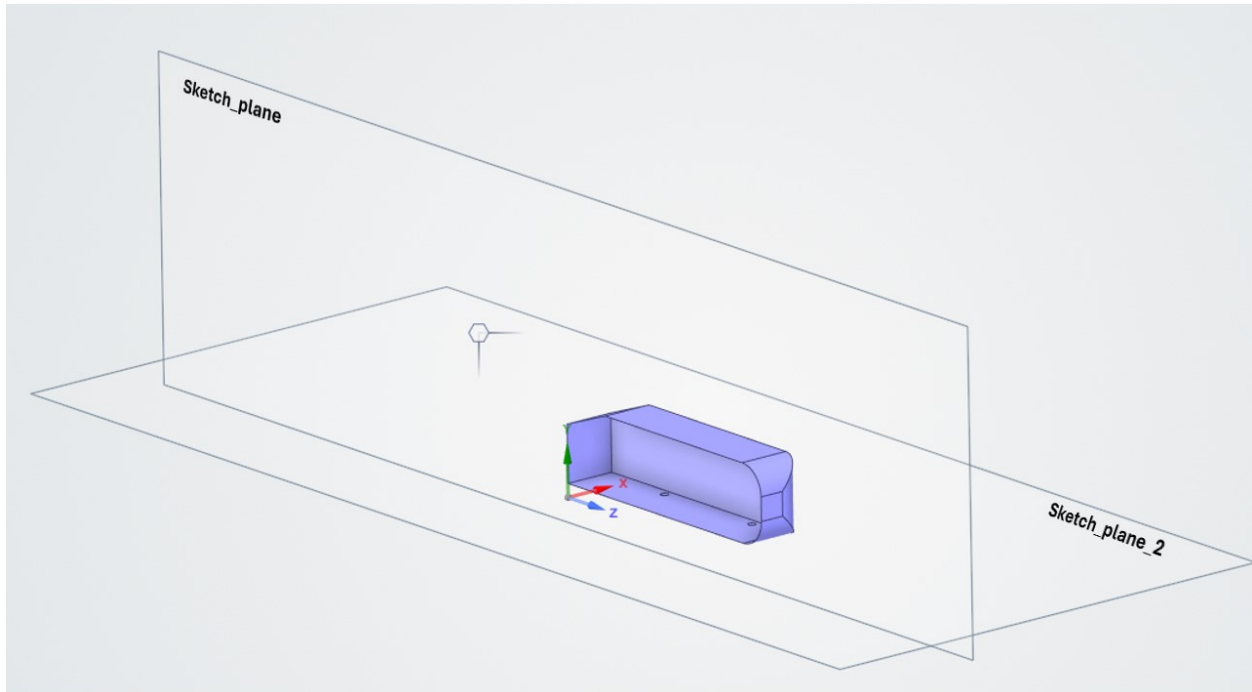
(continued from previous page)

```
warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using
↪ a secure connection.")
```

Create sketches for the Ahmed body, enclosure, and BOI

Define the appropriate sketch planes parallel to the y-z and x-z planes, passing through the origin (namely `sketch_plane` and `sketch_plane_2` respectively).

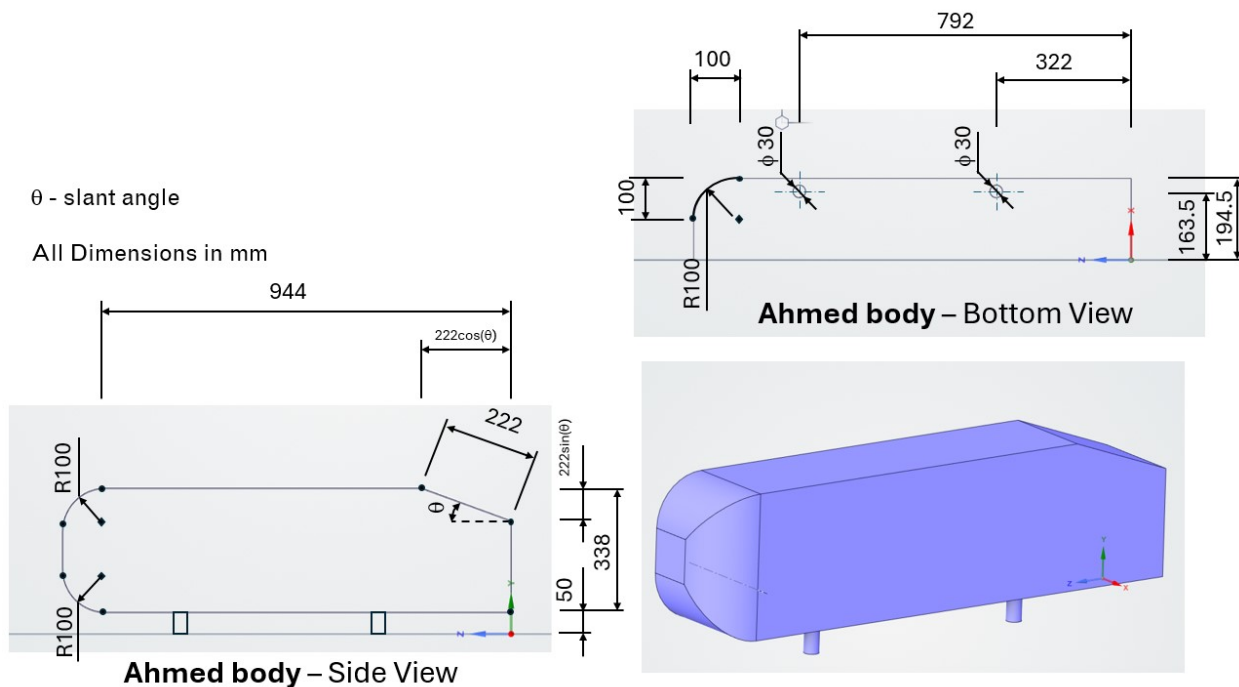
Define the sketch planes



```
[4]: # Define sketch plane on the y-z plane passing through the origin
sketch_plane = Plane(
    origin=Point3D([0, 0, 0]),
    direction_x=UNITVECTOR3D_Y,
    direction_y=UNITVECTOR3D_Z,
)

# Define sketch plane on the x-z plane passing through the origin
sketch_plane_2 = Plane(
    origin=Point3D([0, 0, 0]),
    direction_x=UNITVECTOR3D_X,
    direction_y=UNITVECTOR3D_Z,
)
```


Define the Ahmed body



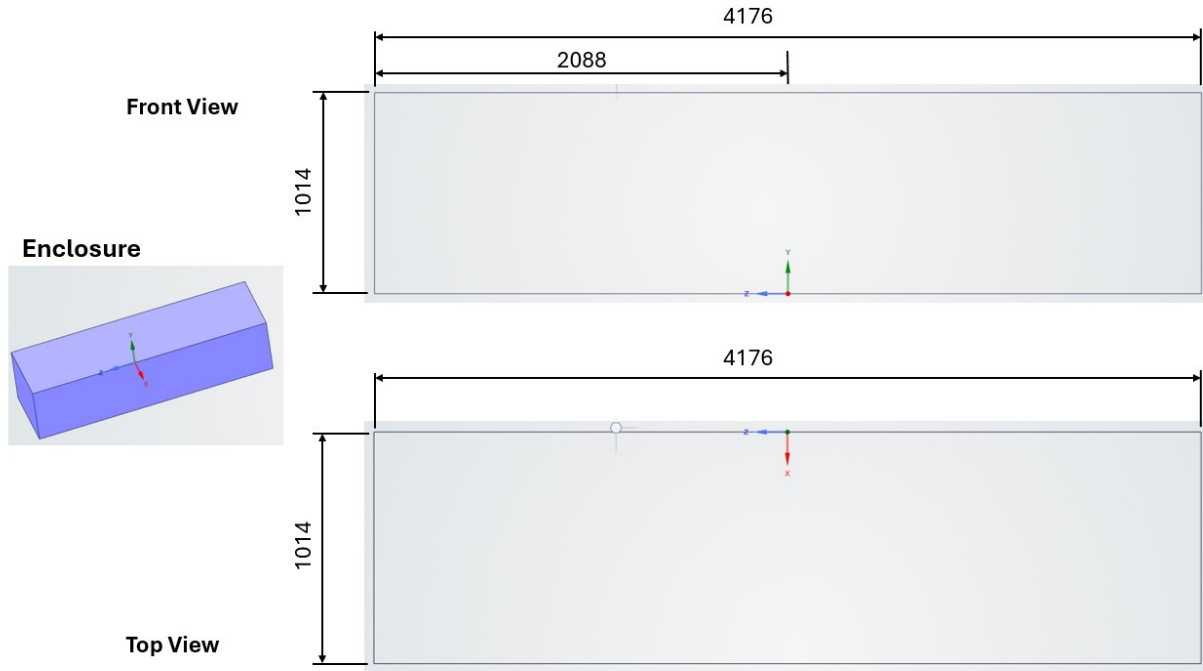
```
[5]: # Calculate the horizontal and vertical distance based on slant angle of 20 degrees
slant_y, slant_x = distance_calculator(hypo=222, slant_angle=20)

# Define sketch for the Ahmed body
ahmed_body_sketch = Sketch(sketch_plane)
ahmed_body_sketch.segment(
    start=Point2D([50, 0]), end=Point2D([338 - slant_y, 0])
).segment_to_point(Point2D([338, slant_x])).segment_to_point(
    Point2D([338, 944])
).arc_to_point(
    end=Point2D([238, 1044]), center=Point2D([238, 944]), clockwise=False
).segment_to_point(
    Point2D([150, 1044])
).arc_to_point(
    end=Point2D([50, 944]), center=Point2D([150, 944]), clockwise=False
).segment_to_point(
    end=Point2D([50, 0])
)
ahmed_body_sketch.plot()
```

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
 ↪ansys/geometry/core/sketch/arc.py:180: PyVistaDeprecationWarning:
 ../../../../.venv/lib/python3.13/site-packages/ansys/geometry/core/sketch/arc.py:180:
 ↪Arguments 'pointa', 'pointb', 'center' must be passed as keyword arguments to function
 ↪'CircularArc'.
 From version 0.50, passing these as positional arguments will result in a TypeError.
 return pv.CircularArc(

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Define the enclosure



```
[6]: # Define sketch for enclosure
enclosure_sketch = Sketch(plane=sketch_plane)
enclosure_sketch.box(center=Point2D([1014 / 2, 0]), height=4176, width=1014)
enclosure_sketch.plot()

# Define sketch for mounting 1
mount_sketch_1 = Sketch(sketch_plane_2)
mount_sketch_1.circle(center=Point2D([163.5, 792]), radius=15)

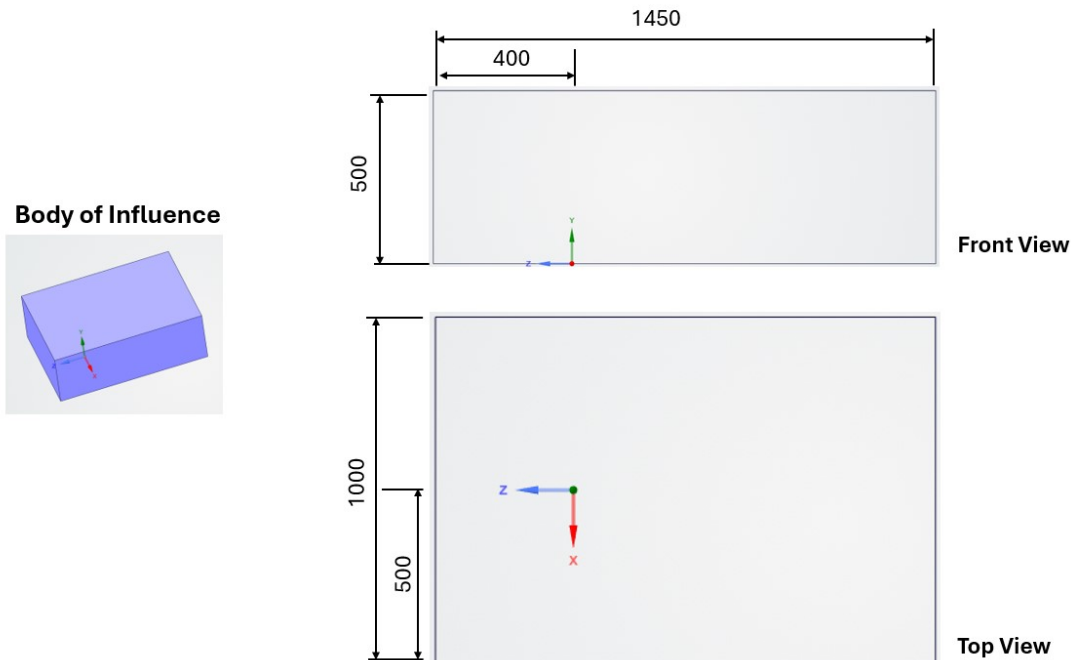
# Define sketch for mounting 2
mount_sketch_2 = Sketch(sketch_plane_2)
mount_sketch_2.circle(center=Point2D([163.5, 322]), radius=15)

# Define sketch for the fillet
ahmed_body_fillet_sketch = Sketch(sketch_plane_2)
ahmed_body_fillet_sketch.segment(
    start=Point2D([194.5, 944]), end=Point2D([194.5, 1044])
).segment_to_point(Point2D([94.5, 1044])).arc_to_point(
    Point2D([194.5, 944]), center=Point2D([94.5, 944]), clockwise=True
)
```

```
EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

```
[6]: <ansys.geometry.core.sketch.sketch.Sketch at 0x7f6410674180>
```

Define the BOI



```
[7]: # Define sketch for BOI
boi_sketch = Sketch(sketch_plane_2)
boi_sketch.box(center=Point2D([0, -325]), width=1000, height=1450)
boi_sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
http-equiv=&quot;Content-...
```

Generate solid bodies from the sketches

From the 2D sketches, generate 3D models by extruding the sketch. First create the design body (namely `ahmed_model`, which is the root part. A component named `Component1` is created under the root part. All the bodies generated as a part of sketch extrusion would be placed within `Component1`.

```
[8]: # Create design object
design = modeler.create_design("ahmed_model")

# Create component
component_1 = design.add_component("Component1")

# Create body `ahmed_body` by extrusion
ahmed_body = component_1.extrude_sketch(
    "ahmed_body", sketch=ahmed_body_sketch, distance=194.5
)

# Create body `ahmed_body_fillet` by cut extrusion
ahmed_body_fillet = component_1.extrude_sketch(
    "ahmed_body_fillet", sketch=ahmed_body_fillet_sketch, distance=-500, cut=True
)
```

(continues on next page)

(continued from previous page)

```

# Create body `enclosure` by extrusion
enclosure = component_1.extrude_sketch(
    "Solid1", sketch=enclosure_sketch, distance=1167
)

# Create body `mounting_1` by extrusion
mount_1 = component_1.extrude_sketch(
    "mount_1", sketch=mount_sketch_1, distance=-100
)

# Create body `mounting_2` by extrusion
mount_2 = component_1.extrude_sketch(
    "mount_2", sketch=mount_sketch_2, distance=-100
)

# Create body `boi` by extrusion
# The direction is negative since the sketch is created in X-Z plane, resulting in the
↪ direction of normal to be parallel to the -Y axis.
boi_body = design.extrude_sketch(
    "boi", sketch=boi_sketch, distance=500, direction="-"
)

```

Perform Boolean operations for region extraction

```

[9]: enclosure.subtract([ahmed_body, mount_1, mount_2])
enclosure.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta
↪ http-equiv="Content-...

```

Group faces and define named selection

```

[10]: plane_surface = []
cylindrical_surface = []

# Group faces of enclosure based on topology
for face in enclosure.faces:
    if face.surface_type.name == "SURFACETYPE_PLANE":
        plane_surface.append(face)
    elif face.surface_type.name == "SURFACETYPE_CYLINDER":
        cylindrical_surface.append(face)

wall_mount = []

# Identify faces associated with mounting
for cyl_face in cylindrical_surface:
    if cyl_face.point(0, 0.5)[1] < 0.050:
        wall_mount.append(cyl_face)

# Identify faces associated with enclosure extremes
outlet_face, inlet_face = face_identifier(faces=plane_surface, axis="z")

```

(continues on next page)

(continued from previous page)

```

symmetry_face, symmetry_x_pos = face_identifier(faces=plane_surface, axis="x")
ground, top = face_identifier(faces=plane_surface, axis="y")

# Create named selection
design.create_named_selection("wall_mount", faces=wall_mount)
design.create_named_selection("inlet", faces=[inlet_face])
design.create_named_selection("outlet", faces=[outlet_face])
design.create_named_selection("wall_ground", faces=[ground])
design.create_named_selection("symmetry_top", faces=[top])
design.create_named_selection("symmetry_center_plane", faces=[symmetry_face])
design.create_named_selection("symmetry_x_pos", faces=[symmetry_x_pos])
design.create_named_selection(
    "ahmed_body_20_0degree_boi_half-boi", faces=boi_body.faces
)

body_planar_surface = list(
    set(plane_surface)
    - set([outlet_face, inlet_face, symmetry_face, symmetry_x_pos, ground, top])
)
body_circular_surface = list(set(cylindrical_surface) - set(wall_mount))

rear_surface, front_surface = face_identifier(faces=body_planar_surface, axis="z")
bottom_surface, top_surface = face_identifier(faces=body_planar_surface, axis="y")
side_surface, symmetry_surface = face_identifier(faces=body_planar_surface, axis="x")

body_circular_surface.append(front_surface)

design.create_named_selection("wall_ahmed_body_front", faces=body_circular_surface)
design.create_named_selection(
    "wall_ahmed_body_main",
    faces=[symmetry_surface, bottom_surface, top_surface],
)

# Identify the face that forms a 20-degree angle with the y-axis.
for face in body_planar_surface:
    if round(math.degrees(math.acos(abs(face.normal().y)))) == 20:
        hypo_face = face
design.create_named_selection(
    "wall_ahmed_body_rear", faces=[hypo_face, rear_surface]
)

```

[10]: ansys.geometry.core.designer.selection.NamedSelection 0x7f63e813c9b0

```

Name          : wall_ahmed_body_rear
Id            : 0:1272
N Bodies      : 0
N Faces       : 2
N Edges       : 0
N Beams       : 0
N Design Points : 0
N Components  : 0
N Vertices    : 0

```

Export model as a PMDB file

Export the geometry into a Fluent-compatible format. The following code exports the geometry into a PMDB file, which retains the named selections.

```
[11]: # Save design
file = design.export_to_pmdb()
print(f"Design saved to {file}")
```

```
Design saved to /home/runner/work/pyansys-geometry/pyansys-geometry/doc/source/examples/
↪04_applied/ahmed_model.pmdb
```

You can import the exported PMDB file into Fluent to set up the mesh and perform the simulation. For an example of how to set up the mesh and boundary conditions in Fluent, see the [Ahmed Body External Aerodynamics Simulation](#) example in the Fluent documentation.

Close session

```
[12]: modeler.close()
```

```
WARNING - localhost:7000 - client - close - Geometry service was not shut down because
↪it was already running...
```

References

[1] S.R. Ahmed, G. Ramm, Some Salient Features of the Time-Averaged Ground Vehicle Wake, SAE-Paper 840300, 1984

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.5 Miscellaneous examples

Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

8.5.1 Miscellaneous: Example template

This example serves as a template for creating new examples in the documentation. It shows developers how to structure their code and comments for clarity and consistency. It also provides a basic outline for importing necessary modules, initializing the modeler, performing operations, and closing the modeler.

It is important to follow the conventions and formatting used in this example to ensure that the documentation is easy to read and understandable.

Example imports

Perform the required imports for this example. This section should include all necessary imports for the example to run correctly.

```
[1]: # Imports
from ansys.geometry.core import launch_modeler
from ansys.geometry.core.math import Point2D
from ansys.geometry.core.sketch import Sketch
```

Initialize the modeler

```
[2]: # Initialize the modeler for this example notebook
m = launch_modeler()
print(m)
```

```
WARNING - - docker_instance - _check_port_availability - Service is already running at_
↪port 700...

Ansys Geometry Modeler (0x7fec162b1160)

Ansys Geometry Modeler Client (0x7fec161e5400)
Target:      localhost:700
Connection: Healthy
Backend info:
  Version:      27.1.0
  Backend type: CORE_LINUX
  Backend number: 20260130.2
  API server number: 1984
  CADIntegration: 27.1.0.7

/home/runner/work/pyansys-geometry/pyansys-geometry/.venv/lib/python3.13/site-packages/
↪ansys/tools/common/cyberchannel.py:188: UserWarning: Starting gRPC client without TLS_
↪on localhost:700. This is INSECURE. Consider using a secure connection.
  warn(f"Starting gRPC client without TLS on {target}. This is INSECURE. Consider using_
↪a secure connection.")
```

Body of your example

Developers can add their code here to perform the desired operations. This section should include comments and explanations to explain what the code is doing.

Example section: Initialize a design

Create a design named example-design.

```
[3]: # Initialize the example design
design = m.create_design("example-design")
```

Example section: Include images

This section demonstrates how to include static images in the documentation. You should place these images in the doc/source/_static/ directory.

You can then reference images in the documentation using the following syntax:



101 Sessions

Example section: Create a sketch and plot it

This section demonstrates how to create a sketch and plot it.

```
[4]: sketch = Sketch()
      sketch.box(Point2D([0, 0]), 10, 10)
      sketch.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Example section: Extrude the sketch and create a body

This section demonstrates how to extrude the sketch and create a body.

```
[5]: design.extrude_sketch(f"BoxBody", sketch, distance=10)
      design.plot()

EmbeddableWidget(value='<iframe srcdoc="<!DOCTYPE html>\n<html>\n  <head>\n    <meta_
↳http-equiv=&quot;Content-...
```

Close the modeler

```
[6]: # Close the modeler
      m.close()

WARNING - localhost:7000 - client - close - Geometry service was not shut down because_
↳it was already running...
```


Download this example

Download this example as a [Jupyter Notebook](#) or as a [Python script](#).

CONTRIBUTE

Overall guidance on contributing to a PyAnsys library appears in the [Contributing](#) topic in the *PyAnsys Developer's Guide*. Ensure that you are thoroughly familiar with this guide before attempting to contribute to PyAnsys Geometry.

The following contribution information is specific to PyAnsys Geometry.

9.1 Clone the repository

To clone and install the latest PyAnsys Geometry release in development mode, run these commands:

```
git clone https://github.com/ansys/pyansys-geometry
cd pyansys-geometry
python -m pip install --upgrade pip
pip install -e . --group dev
```

9.2 Post issues

Use the [PyAnsys Geometry Issues](#) page to submit questions, report bugs, and request new features. When possible, you should use these issue templates:

- Bug, problem, error: For filing a bug report
- Documentation error: For requesting modifications to the documentation
- Adding an example: For proposing a new example
- New feature: For requesting enhancements to the code

If your issue does not fit into one of these template categories, you can click the link for opening a blank issue.

To reach the project support team, email pyansys.core@ansys.com.

9.3 View documentation

Documentation for the latest stable release of PyAnsys Geometry is hosted at [PyAnsys Geometry Documentation](#).

In the upper right corner of the documentation's title bar, there is an option for switching from viewing the documentation for the latest stable release to viewing the documentation for the development version or previously released versions.

9.4 Adhere to code style

PyAnsys Geometry follows the PEP8 standard as outlined in [PEP 8](#) in the *PyAnsys Developer's Guide* and implements style checking using [pre-commit](#).

To ensure your code meets minimum code styling standards, run these commands:

```
pip install pre-commit
pre-commit run --all-files
```

You can also install this as a pre-commit hook by running this command:

```
pre-commit install
```

This way, it's not possible for you to push code that fails the style checks:

```
$ pre-commit install
$ git commit -am "added my cool feature"
ruff.....Passed
codespell.....Passed
check for merge conflicts.....Passed
debug statements (python).....Passed
check yaml.....Passed
trim trailing whitespace.....Passed
Validate GitHub Workflows.....Passed
check pre-commit.ci config.....Passed
```

9.5 Build the documentation

Note

To build the documentation, you must have the Geometry Service installed and running on your machine because it is used to generate the examples in the documentation. It is also recommended that the service is running as a Docker container.

If you do not have the Geometry Service installed, you can still build the documentation, but the examples are not generated. To build the documentation without the examples, define the following environment variable:

```
# On Linux or macOS
export BUILD_EXAMPLES=false

# On Windows CMD
set BUILD_EXAMPLES=false

# On Windows PowerShell
$env:BUILD_EXAMPLES="false"
```

To build the documentation locally, you must run this command to install the documentation dependencies:

```
pip install -e . --group doc
```

Then, navigate to the docs directory and run this command:

```
# On Linux or macOS
make html

# On Windows
./make.bat html
```

The documentation is built in the docs/_build/html directory.

You can clean the documentation build by running this command:

```
# On Linux or macOS
make clean

# On Windows
./make.bat clean
```

9.6 Adding examples

Users can collaborate with examples to this documentation by adding new examples. A reference commit of the changes that adding an example requires is shown here:

<https://github.com/ansys/pyansys-geometry/pull/1454/commits/7fcf02f86f05e0e5ce1c1071c3c5fcd274ec481c>

To add a new example, follow these steps:

1. Create a new notebook in the doc/source/examples directory, under the appropriate folder for your example.
2. Use the doc\source\examples\99_misc\template.mystnb file as a reference for creating your example notebook. It contains the necessary metadata and structure for a PyAnsys Geometry example.
3. Add the new notebook to the doc/source/examples.rst file.
4. Store a thumbnail image of the example in the doc/source/_static/thumbnails directory.
5. Link the thumbnail image to your example file in the doc/source/conf.py file as shown in the reference commit.

You can also test the correct build process of a new example by performing the following steps:

1. Run the following command to install the documentation dependencies:

```
pip install -e . --group doc
```

2. Navigate to the doc directory and run the following command:

```
# On Linux or macOS
make single-example examples/01_getting_started/01_math.mystnb

# On Windows
./make.bat single-example examples/01_getting_started/01_math.mystnb
```

Note

The example name must be the same as the notebook name, with its path starting at the examples directory.

3. Check the doc/source/_build/html directory for the generated documentation and open the index.html file in your browser.

9.7 Run tests

PyAnsys Geometry uses `pytest` for testing.

9.7.1 Prerequisites

Prior to running the tests, you must run this command to install the test dependencies:

```
pip install -e . --group tests
```

Make sure to define the port and host of the service using the following environment variables:

```
# On Linux or macOS
export ANSRV_GEO_PORT=5000
export ANSRV_GEO_HOST=localhost

# On Windows CMD
set ANSRV_GEO_PORT=5000
set ANSRV_GEO_HOST=localhost

# On Windows PowerShell
$env:ANSRV_GEO_PORT=5000
$env:ANSRV_GEO_HOST="localhost"
```

9.7.2 Running the tests

To run the tests, navigate to the root directory of the repository and run this command:

```
pytest
```

Note

The tests require the Geometry Service to be installed and running on your machine. The tests fail if the service is not running. It is expected for the Geometry Service to be running as a Docker container.

If you do not have the Geometry Service running as a Docker container, but you have it running on your machine, you can still run the tests with the following argument:

```
pytest --use-existing-service=yes
```

In this section, users are able to download a set of assets related to PyAnsys Geometry.

10.1 Documentation

The following links provide users with downloadable documentation in various formats

- [Documentation in HTML format](#)
- [Documentation in PDF format](#)

10.2 Wheelhouse

If you lack an internet connection on your installation machine, you should install PyAnsys Geometry by downloading the wheelhouse archive.

Each wheelhouse archive contains all the Python wheels necessary to install PyAnsys Geometry from scratch on Windows, Linux, and MacOS from Python 3.10 to 3.13. You can install this on an isolated system with a fresh Python installation or on a virtual environment.

For example, on Linux with Python 3.10, unzip the wheelhouse archive and install it with:

```
unzip ansys-geometry-core-v0.15.dev0-all-wheelhouse-ubuntu-latest-3.10.zip wheelhouse
pip install ansys-geometry-core -f wheelhouse --no-index --upgrade --ignore-installed
```

If you are on Windows with Python 3.10, unzip to a wheelhouse directory by running `-d wheelhouse` (this is required for unzipping to a directory on Windows) and install using the preceding command.

Consider installing using a [virtual environment](#).

The following wheelhouse files are available for download:

10.2.1 Linux

- [Linux wheelhouse for Python 3.10](#)
- [Linux wheelhouse for Python 3.11](#)
- [Linux wheelhouse for Python 3.12](#)
- [Linux wheelhouse for Python 3.13](#)

10.2.2 Windows

- Windows wheelhouse for Python 3.10
- Windows wheelhouse for Python 3.11
- Windows wheelhouse for Python 3.12
- Windows wheelhouse for Python 3.13

10.2.3 MacOS

- MacOS wheelhouse for Python 3.10
- MacOS wheelhouse for Python 3.11
- MacOS wheelhouse for Python 3.12
- MacOS wheelhouse for Python 3.13

10.3 Geometry service Docker container assets

Build the latest Geometry service Docker container using the following assets. For information on how to build the container, see *Docker containers*.

Currently, the Geometry service backend is mainly delivered as a **Windows** Docker container. However, these containers require a Windows machine to run them.

10.3.1 Windows container

Note

Only Ansys employees with access to <https://github.com/ansys/pyansys-geometry-binaries> can download these binaries.

- Latest Geometry service binaries for Windows containers
- Latest Geometry service Dockerfile for Windows containers

RELEASE NOTES

This document contains the release notes for the PyAnsys Geometry project.

11.1 0.14.2 - January 07, 2026

Added

Tracking updates	#2359
Feat: intersect curves and surfaces	#2461
NURBS warnings for backend < 26.1	#2487

Dependencies

Bump notebook from 7.4.7 to 7.5.0 in the docs-deps group	#2424
Bump ansys-tools-common from 0.3.0 to 0.3.1	#2426
Bump nbsphinx from 0.9.6 to 0.9.8 in the docs-deps group	#2428
Bump pre-commit from 4.4.0 to 4.5.0	#2429
Bump beartype from 0.22.6 to 0.22.7	#2435
Bump ansys-api-discovery from 1.0.14 to 1.0.15	#2436
Build: bump ansys-api-discovery from 1.0.15 to 1.0.16	#2442
Build: bump ansys-api-discovery from 1.0.16 to 1.0.17	#2447
Build: bump pytest-pyvista from 0.3.1 to 0.3.2	#2448
Build: bump beartype from 0.22.7 to 0.22.8	#2449
Build: bump numpydoc from 1.9.0 to 1.10.0 in the docs-deps group	#2452
Build: bump pytest from 8.4.2 to 9.0.0	#2453
Build: bump ansys-tools-common from 0.3.1 to 0.4.0	#2457
Build: bump pytest from 9.0.0 to 9.0.2	#2458
Build: bump ansys-api-discovery from 1.0.17 to 1.0.18	#2462
Build: bump ansys-sphinx-theme[autoapi] from 1.6.3 to 1.6.4 in the docs-deps group	#2465
Build: bump ansys-api-discovery from 1.0.18 to 1.0.19	#2466
Build: bump matplotlib from 3.10.7 to 3.10.8	#2467
Bump beartype from 0.22.8 to 0.22.9	#2470
Bump notebook from 7.5.0 to 7.5.1 in the docs-deps group	#2477
Bump pre-commit from 4.5.0 to 4.5.1	#2479
Bump numpy from 2.3.3 to 2.4.0	#2489
Bump pint from 0.24.4 to 0.25.2	#2490
Bump scipy from 1.16.2 to 1.16.3	#2491

Documentation

Update contributing to ref pyansys geometry	#2440
Fix duplicate docstrings	#2485

Fixed

Skip tessellation transform if identity matrix	#2438
Fix: v1 imprinting and projecting curves fixes	#2459
Fix: doc string typo	#2469
Docker launch method requires proper network resolution	#2493

Maintenance

update CHANGELOG for v0.9.0	#1753
Chore: v1 implementation of bodies stub	#2400
V1 implementation of patterns stub	#2416
V1 implementation of prepare tools	#2417
Update CHANGELOG for v0.14.1	#2425
Update missing or outdated files	#2427
Bump github/codeql-action from 4.31.5 to 4.31.6 in the actions group	#2430
Pre-commit automatic update	#2431, #2481, #2486
V1 implementation of parts and commands stubs	#2432
V1 implementation of named selections stub	#2433
Implement Points V1	#2434
Ci: bump the actions group with 2 updates	#2437
Chore: v1 implementation of repair stubs	#2441
Chore: v1 implementation of file/designs stub	#2443
Chore: v1 implementation of beam stub	#2444
Chore: v1 implementation of curves stub	#2445
Chore: v1 Implementation of driving dimensions stub	#2446
Chore: pre-commit automatic update	#2450, #2468
Chore: v1 fix imprint curves	#2454
Chore: v1 general cleanup	#2455
Chore: fix repair/prepare v1 issues	#2456
Ci: enable v1 testing on pull request	#2464
Bump ansys/actions from 10.2.0 to 10.2.3 in the actions group	#2471
Include tracker testing with v1	#2473
Test v1 on unstable image promotion	#2474
Bump codecov/codecov-action from 5.5.1 to 5.5.2 in the actions group	#2475
Bump actions/cache from 4.3.0 to 5.0.1 in the actions group	#2478
Moving v1 testing to main pipeline	#2482
Bump the actions group across 1 directory with 3 updates	#2483
Remove pragma statements and upload v1 coverage	#2484

Test

Test: adjusting small face count	#2463
----------------------------------	-------

11.2 0.14.1 - November 26, 2025

Documentation

Improve secure connection communication	#2423
---	-------

Maintenance

Update CHANGELOG for v0.14.0	#2421
Bump dev version to 0.15.dev0	#2422

11.3 0.14.0 - November 26, 2025

Added

Sweepable body detection	#2394
Secure grpc channels	#2411
Enhancements to transport mode	#2419

Dependencies

Bump beartype from 0.22.5 to 0.22.6	#2406
Bump ansys-api-discovery from 1.0.9 to 1.0.10	#2407
Bump ansys-api-discovery from 1.0.10 to 1.0.14	#2414

Maintenance

V1 implementation of components stub	#2401
V1 implementation of coordinate system stub	#2402
Update CHANGELOG for v0.13.0	#2404
Bump dev version	#2405
Bump github/codeql-action from 4.31.4 to 4.31.5 in the actions group	#2408
Pre-commit automatic update	#2409
V1 implementation of materials stub	#2410
V1 implementation of faces + edges stubs	#2412
V1 implementation of measurement tools and model tools stubs	#2413
Bump actions/setup-python from 6.0.0 to 6.1.0 in the actions group	#2415
Cleanup of deprecated arguments and methods	#2420

11.4 0.13.0 - November 24, 2025

Added

Preserve original signature of decorated methods	#2346
Set current working dir for linux	#2349
Expose proper server working directory	#2350
Add Named Selections query to geometric entities	#2356
Properly handling v0 and v1 initialization	#2368
Adding ``py.typed`` file	#2372
Rename named selection	#2382
Expose enclosure methods	#2383
Transfer named selections for special boolean subtract	#2392
V1 unsupported stub	#2397

Dependencies

Bump jupyter from 1.17.3 to 1.18.1 in the docs-deps group	#2342
Bump beartype from 0.22.2 to 0.22.3	#2344
Bump beartype from 0.22.3 to 0.22.4	#2347
Bump pyvista[jupyter] from 0.46.3 to 0.46.4	#2353
Bump beartype from 0.22.4 to 0.22.5	#2355
Bump ansys-api-geometry from 0.4.83 to 0.4.84	#2358
Switch to ansys-api-discovery for protos package	#2365
Bump ansys-api-discovery from 1.0.5 to 1.0.6	#2370
Migrating to ansys-tools-common	#2373
Bump quarto-cli from 1.8.25 to 1.8.26	#2377
Bump pre-commit from 4.3.0 to 4.4.0	#2378
Bump ansys-api-discovery from 1.0.6 to 1.0.7	#2379
Bump ansys-api-discovery from 1.0.7 to 1.0.8	#2385
Bump ansys-api-discovery from 1.0.8 to 1.0.9	#2398

Fixed

Logo removal height issues	#2357
Change find split edges default	#2361
Add axis for circular pattern	#2367

Maintenance

update CHANGELOG for v0.9.0	#1753
Update CHANGELOG for v0.12.0	#2334
Pre-commit automatic update	#2335, #2345, #2354, #2362, #2376
Update CHANGELOG for v0.12.1	#2339
Add unit support	#2340
Bump the actions group with 3 updates	#2343
Bump github/codeql-action from 4.31.0 to 4.31.2 in the actions group	#2348
Bump ansys/actions from 10.1.4 to 10.1.5 in the actions group	#2351
Update missing or outdated files	#2352
Update angle docstring	#2360
Quarantine tessellation tests	#2363
Remove print statements	#2364
Bump github/codeql-action from 4.31.2 to 4.31.3 in the actions group	#2375
Bump the actions group across 1 directory with 2 updates	#2381
Adding testing for v1 protos	#2389
Update SECURITY.md	#2390
V1 implementation of admin and assembly condition stubs	#2391
Bump actions/checkout from 5.0.1 to 6.0.0 in the actions group	#2393, #2399
Dbu application to commands script v1	#2395
Implement v1 for conversions.py	#2403

Test

Fix tessellation tests	#2341
Re-enable tessellation tests	#2369

11.5 0.12.1 - October 21, 2025

Fixed

Decorator order issue	#2337
-----------------------	-------

11.6 0.12.0 - October 20, 2025

Added

Tracking boolean operations	#2153
Helix detection	#2232
Move geometry commands to versioned architecture	#2234
Pull commands pt. 1	#2241
Loft profiles with guides	#2252
Check for match	#2255
Grpc rearchitecture - components model	#2263
Cleaning up usage of unsupported stub	#2267
NURBS surface body creation	#2273
Combine and merge bodies	#2282
Tessellation speed enhancements	#2294
Implement version based import	#2307
Return backend info on repr for client and modeler	#2311
Finalize version handling for design class	#2323
Edge tessellations, stream full design tessellation	#2328

Dependencies

Bump ansys-api-geometry from 0.4.74 to 0.4.75	#2236
Bump quarto-cli from 1.7.34 to 1.8.24	#2242
Bump panel from 1.8.0 to 1.8.1	#2243
Bump vtk from 9.5.1 to 9.5.2	#2245
Bump ansys-api-geometry from 0.4.75 to 0.4.76	#2246
Bump ansys-api-geometry from 0.4.76 to 0.4.77	#2253
Upgrade grpcio and protobuf	#2258
Bump notebook from 7.4.5 to 7.4.6 in the docs-deps group	#2268
Bump pyyaml from 6.0.2 to 6.0.3	#2271
Bump notebook from 7.4.6 to 7.4.7 in the docs-deps group	#2275
Bump quarto-cli from 1.8.24 to 1.8.25	#2281
Bump-ansys-tools-viz-interface	#2283
Bump ansys-api-geometry from 0.4.78 to 0.4.80	#2290
Bump pytest-cov from 6.3.0 to 7.0.0	#2299
Bump panel from 1.8.1 to 1.8.2	#2304
Bump trame-vtk from 2.9.1 to 2.10.0	#2305
Bump ansys-sphinx-theme	#2308
Bump beartype from 0.21.0 to 0.22.2	#2314
Bump matplotlib from 3.10.6 to 3.10.7	#2315
Bump scipy (1.16.2) and numpy (2.3.3)	#2318
Bump pytest-pyvista from 0.2.0 to 0.3.1	#2322
Bump ansys-api-geometry from 0.4.81 to 0.4.82	#2327

Fixed

Revert skipped test for test_plot_design_face_colors	#2266
Card message on failure	#2274
Empty tessellation data conversion	#2284
Turn off knot normalization to match SC	#2300
Bool command implementation	#2303
Fix copy with occurrence	#2320
Re enable logs testing	#2321
Disable broken design tessellation test	#2324

Maintenance

Update CHANGELOG for v0.11.3	#2235
Bump ansys/actions from 10.1.0 to 10.1.1 in the actions group	#2237
Pre-commit automatic update	#2247, #2269, #2285, #2319
Adapt docker run commands	#2249
Bump ansys/actions from 10.1.1 to 10.1.2 in the actions group	#2250
Bump actions/cache from 4.2.4 to 4.3.0 in the actions group	#2254
Backwards compatibility changes on Docker images	#2256
Bump github/codeql-action from 3.30.3 to 3.30.4 in the actions group	#2259
Bump the actions group with 3 updates	#2270, #2280
Cleanup miscellaneous grpc refactors	#2272
Zizmor fixes	#2277
Bump ansys/actions from 10.1.3 to 10.1.4 in the actions group	#2278
Remove tracker testing	#2279
Run on GitHub runners directly	#2286
Removing commands stub from body module	#2288
Bump github/codeql-action from 3.30.6 to 4.30.7 in the actions group	#2291
Prevent invalid kwargs from being passed to launcher methods	#2293
Remove grpc ids	#2295
Update dependabot configuration	#2297
Dependabot cooldown on github-actions is not supported	#2298
Add write permissions to doc deploy jobs	#2302
Bump github/codeql-action from 4.30.7 to 4.30.8 in the actions group	#2306
Skip flaky test with retrieval of service logs	#2309
Adding YAML and TOML formatting hooks	#2312
Using group dependencies	#2316
Cleanup usage of EntityIdentifier	#2317
Bump github/codeql-action from 4.30.8 to 4.30.9 in the actions group	#2329
Versioning fixes	#2330

Test

Verifying issue 2251 - double import and export to stride fails	#2257
Adding test coverage for nurbs, design, and geometry commands	#2260
Skip new failures from nuget updated	#2262
Adding lines to coverage code implemented for nurbs surface creation	#2301

11.7 0.11.3 - September 17, 2025

Added

Move_imprint_edges, offset_edges, draft_faces, thicken_faces implementations	#2214
NURBS surface implementation	#2222
For 261 dotnet 8 is included in core service	#2226

Dependencies

Bump pyvista[jupyter] from 0.46.2 to 0.46.3	#2203
Bump jupyter from 1.17.2 to 1.17.3 in the docs-deps group	#2206
Bump vtk from 9.5.0 to 9.5.1	#2208
Bump quarto-cli from 1.7.33 to 1.7.34	#2209
Bump matplotlib from 3.10.5 to 3.10.6	#2211
Bump ansys-sphinx-theme[autoapi] from 1.5.3 to 1.6.0 in the docs-deps group	#2213
Bump pytest from 8.4.1 to 8.4.2	#2218
Bump pytest-cov from 6.2.1 to 6.3.0	#2227
Bump ansys-sphinx-theme[autoapi] from 1.6.0 to 1.6.1 in the docs-deps group	#2228
Bump panel from 1.7.5 to 1.8.0	#2233

Documentation

Update html_context with PyAnsys tags	#2200
---------------------------------------	-------

Fixed

Add skip_if_discovery method	#2202
------------------------------	-------

Maintenance

Re-enable 25.1 testing	#2185
Update CHANGELOG for v0.11.2	#2197
Pre-commit automatic update	#2205, #2219, #2230
Bump the actions group with 2 updates	#2207
Bump ansys/actions from 10.0.15 to 10.0.16 in the actions group	#2212
Bump the actions group with 4 updates	#2215
Bump the actions group with 3 updates	#2216
Bump github/codeql-action from 3.30.1 to 3.30.2 in the actions group	#2221
Bump github/codeql-action from 3.30.2 to 3.30.3 in the actions group	#2225
Bump ansys/actions from 10.0.20 to 10.1.0 in the actions group	#2229

Test

Adding test for vertex and tests based on spaceclaim tutorials	#2157
Adding test for nurbs sketching	#2193
New test and remove skip	#2199

11.8 0.11.2 - August 26, 2025

Added

Option to write body facets on save	#2169
Body sweep_with_guide	#2179
Remove materials	#2180

Dependencies

Bump ansys-api-geometry from 0.4.71 to 0.4.72	#2175
Bump pyvista[jupyter] from 0.46.0 to 0.46.1	#2176
Bump ansys-api-geometry from 0.4.72 to 0.4.73	#2177
Bump ansys-api-geometry from 0.4.73 to 0.4.74	#2183
Bump requests from 2.32.4 to 2.32.5	#2192
Bump pyvista[jupyter] from 0.46.1 to 0.46.2	#2196

Fixed

Make import component named selections optional	#2144
Prevent tool and command classes instantiation outside of modeler object	#2188
Mathematical approximations leading to potential issues - allowing rounding	#2190

Maintenance

Update CHANGELOG for v0.11.1	#2173
Enable timeout for pytest to prevent blockage	#2174
Pre-commit automatic update	#2181, #2195
Bump github/codeql-action from 3.29.9 to 3.29.10 in the actions group	#2182
Bump codecov/codecov-action from 5.4.3 to 5.5.0 in the actions group	#2186
Bump github/codeql-action from 3.29.10 to 3.29.11 in the actions group	#2191

11.9 0.11.1 - August 13, 2025

Added

Nurbs sketching and surface support	#2104
Surface body unit test fix	#2118
Verifying backwards compatibility	#2124
Component operations - make_independent() and import_named_selections()	#2129
Set multiple export ids	#2148

Dependencies

Bump vtk from 9.4.2 to 9.5.0	#2082
Bump ansys-api-geometry from 0.4.65 to 0.4.66	#2117
Bump ansys-api-geometry from 0.4.66 to 0.4.67	#2123
Bump pyvista[jupyter] from 0.45.2 to 0.45.3	#2126
Bump ansys-api-geometry from 0.4.67 to 0.4.68	#2131
Bump trame-vtk from 2.9.0 to 2.9.1	#2134
Bump panel from 1.7.4 to 1.7.5	#2135
Bump pygltflib from 1.16.4 to 1.16.5	#2139
Bump matplotlib from 3.10.3 to 3.10.5	#2151
Bump ansys-api-geometry from 0.4.68 to 0.4.69	#2154
Bump notebook from 7.4.4 to 7.4.5 in the docs-deps group	#2160
Bump quarto-cli from 1.7.32 to 1.7.33	#2165
Bump pyvista[jupyter] from 0.45.3 to 0.46.0	#2171
Bump ansys-api-geometry from 0.4.69 to 0.4.71	#2172

Fixed

Workflow 2b nurbs fixes	#2137
Clarify logic in min distance grpc call	#2166

Maintenance

Update changelog for v0.11.0	#2116
Bump minor version in main	#2119
Pre-commit automatic update	#2125, #2140, #2150, #2163
Bump github/codeql-action from 3.29.2 to 3.29.3 in the actions group	#2127
Bump github/codeql-action from 3.29.3 to 3.29.4 in the actions group	#2130
Bump ansys/actions from 10.0.12 to 10.0.13 in the actions group	#2133
Bump the actions group across 1 directory with 5 updates	#2158
Bump the actions group with 2 updates	#2164
Remove temporarily 25.1 backwards compatibility checks	#2168
Bump github/codeql-action from 3.29.8 to 3.29.9 in the actions group	#2170

Test

Adding test coverage for modeler, plotting, logger, and parameters	#2108
Internalize external documents	#2109
Add fixture for running with tracker	#2120
Import named selection tests	#2138
Nurbs length test	#2142
Temp skip	#2147
Skip x_t reimport tests	#2161

11.10 0.11.0 - July 15, 2025

Added

Update with delta	#1922
Find and fix stitch/missing/small faces enhancements	#1953
Nurbs and trimmedcurve enhancements	#1994
Allow logos linux 26 1	#2048
Nurbscurve conversions	#2053
Add components to ns	#2068
Add mating conditions (align, tangent, orient)	#2069
Vertex references	#2083

Dependencies

Bump jupyter from 1.17.1 to 1.17.2 in the docs-deps group	#2024
Bump ansys-tools-visualization-interface from 0.9.1 to 0.9.2	#2027
Bump trame-vtk from 2.8.15 to 2.8.17	#2028
Bump pytest from 8.3.5 to 8.4.0	#2034
Bump requests from 2.32.3 to 2.32.4	#2040
Bump beartype from 0.20.2 to 0.21.0	#2042
Bump panel from 1.6.1 to 1.7.1	#2043
Bump ansys-tools-path from 0.7.1 to 0.7.2	#2044
Bump quarto-cli from 1.7.31 to 1.7.32	#2050
Bump ansys-tools-path from 0.7.2 to 0.7.3	#2051
Bump pytest-cov from 6.1.1 to 6.2.1	#2052
Limiting ansys-tools-visualization-interface	#2054
Bump pytest from 8.4.0 to 8.4.1	#2065
Bump panel from 1.7.1 to 1.7.2	#2077
Bump trame-vtk from 2.8.17 to 2.9.0	#2078
Bump numpydoc from 1.8.0 to 1.9.0 in the docs-deps group	#2080
Bump ansys-api-geometry from 0.4.62 to 0.4.64	#2081
Bump ansys-api-geometry from 0.4.64 to 0.4.65	#2085
Bump notebook from 7.4.3 to 7.4.4 in the docs-deps group	#2086
Bump ansys-sphinx-theme[autoapi] from 1.5.2 to 1.5.3 in the docs-deps group	#2089
Bump panel from 1.7.2 to 1.7.4	#2112
Bump pytest-pyvista from 0.1.9 to 0.2.0	#2113

Documentation

Adding extra line	#2026
Add proper disclaimer to binaries repository	#2060
Add warning section for minimum version on methods	#2062
Add deepwiki link	#2073

Fixed

Make sure export_glb is handling a single polydata object	#2032
Prevent the creation of empty named selections	#2063
Revert visualization changes	#2084
Internalize document after insert: update test	#2092

Maintenance

Update changelog for v0.10.9	#2023
Bump ansys/actions from 10.0.4 to 10.0.6 in the actions group	#2025
Bump ansys/actions from 10.0.6 to 10.0.8 in the actions group	#2029
Pre-commit automatic update	#2033, #2066, #2076, #2090, #2111
Bump ansys/actions from 10.0.8 to 10.0.9 in the actions group	#2035
Bump ansys/actions from 10.0.9 to 10.0.10 in the actions group	#2038
Bump the actions group with 2 updates	#2041
Upload code coverage on linux	#2049
Bump ansys/actions from 10.0.11 to 10.0.12 in the actions group	#2071
Bump github/codeql-action from 3.29.0 to 3.29.1 in the actions group	#2075
Bump github/codeql-action from 3.29.1 to 3.29.2 in the actions group	#2079
Using proper artifactory location	#2114

Test

Expand code coverage and fix a few things	#2039
Add more tests and update some tests	#2046
Expanding test coverage for sketch	#2047
Expanding test coverage for designer and math	#2061
Adding test coverage for designer, sketch, misc	#2070
Add more tests to expand coverage	#2087
Add stride named selection import test	#2088
Add more code coverage	#2096
Logo removal should work on linux now	#2098
Expand coverage and add bug fix test	#2103
Bug fix test and round trip open file tests	#2107

11.11 0.10.9 - June 05, 2025

Dependencies

bump ansys-sphinx-theme[autoapi] from 1.4.4 to 1.4.5 in the docs-deps group	#2008
bump ansys-sphinx-theme[autoapi] from 1.4.5 to 1.5.0 in the docs-deps group	#2009
bump notebook from 7.4.2 to 7.4.3 in the docs-deps group	#2010
bump geomdl from 5.3.1 to 5.4.0	#2012
bump ansys-sphinx-theme[autoapi] from 1.5.0 to 1.5.2 in the docs-deps group	#2014

Fixed

Typo in the open request construction	#2022
---------------------------------------	-------

Maintenance

update CHANGELOG for v0.10.8	#2006
bump ansys/actions from 9.0.12 to 9.0.13 in the actions group	#2011
pre-commit automatic update	#2015
improving CodeQL	#2016
fix labeler permissions	#2017
Bump ansys/actions from 9.0.13 to 10.0.1 in the actions group	#2018
Bump the actions group with 2 updates	#2019, #2021

Test

Update Reader support info and add more import tests	#2013
--	-------

11.12 0.10.8 - May 27, 2025**Added**

repair tools refactoring	#1912
use license metadata properly	#1961
add 261 version api versions list	#1980
grpc reachitecture - several modules	#1988
deprecating product_version in favor of version	#1998

Dependencies

bump quarto-cli from 1.6.42 to 1.7.29	#1962
bump jupyter from 1.16.7 to 1.17.1 in the docs-deps group	#1963
bump ansys-sphinx-theme[autoapi] from 1.4.2 to 1.4.3 in the docs-deps group	#1967
bump ansys-api-geometry from 0.4.58 to 0.4.59	#1968
bump ansys-sphinx-theme[autoapi] from 1.4.3 to 1.4.4 in the docs-deps group	#1969
bump notebook from 7.4.1 to 7.4.2 in the docs-deps group	#1971
bump quarto-cli from 1.7.29 to 1.7.30	#1972
bump matplotlib from 3.10.1 to 3.10.3	#1974
bump pyvista[jupyter] from 0.45.0 to 0.45.1	#1975
bump scipy from 1.15.2 to 1.15.3	#1976
bump pyvista[jupyter] from 0.45.1 to 0.45.2	#1981
bump ansys-api-geometry from 0.4.59 to 0.4.60	#1987
bump quarto-cli from 1.7.30 to 1.7.31	#1991
bump ansys-api-geometry from 0.4.60 to 0.4.61	#1992
bump numpy from 2.2.5 to 2.2.6	#1995
bump ansys-api-geometry from 0.4.61 to 0.4.62	#2003

Documentation

change python3statement url	#1965
-----------------------------	-------

Fixed

myst warning – all cells must be of same type	#1970
---	-------

Maintenance

update CHANGELOG for v0.10.7	#1959
pre-commit automatic update	#1964, #1978, #1993, #2004
bump ansys/actions from 9.0.7 to 9.0.8 in the actions group	#1966
bump ansys/actions from 9.0.8 to 9.0.9 in the actions group	#1973
bump ansys/actions from 9.0.9 to 9.0.19 in the actions group	#1979
bump ansys/actions from 9.0.19 to 9.0.20 in the actions group	#1982
bump ansys/actions from 9.0.20 to 9.0.21 in the actions group	#1983
bump ansys/actions from 9.0.21 to 9.0.22 in the actions group	#1984
revert ansys/actions release	#1985
bump the actions group with 2 updates	#1990
bump ansys/actions from 9.0.9 to 9.0.11 in the actions group	#1996
fix the usage of unstable container skip	#2001
bump ansys/actions from 369ef11a9888875682d1a6b0ec65f82c4d8a664d to 5dc39c7838f50142138f7ac518ff3e4dca065d97 in the actions group	#2002

11.13 0.10.7 - May 05, 2025

Added

grpc driving dimensions stub implementation	#1921
move coordinate systems stub to grpc layer	#1943

Dependencies

bump numpy from 2.2.4 to 2.2.5	#1935
bump pygltflib from 1.16.3 to 1.16.4	#1940
bump notebook from 7.3.3 to 7.4.1 in the docs-deps group	#1946
bump ansys-api-geometry from 0.4.57 to 0.4.58	#1954

Documentation

Update CONTRIBUTORS.md with the latest contributors	#1938
ignore stackoverflow link	#1957

Fixed

core service launcher missing CADIntegration bin folder in path	#1958
---	-------

Maintenance

update CHANGELOG for v0.10.6	#1933
use v4 of pyvista/setup-headless-display-action	#1934
bump github/codeql-action from 3.28.15 to 3.28.16 in the actions group	#1936
bump the actions group with 2 updates	#1937, #1942
pre-commit automatic update	#1941
bump github/codeql-action from 3.28.16 to 3.28.17 in the actions group	#1956

11.14 0.10.6 - April 22, 2025**Added**

grpc prepare tools stub implementation	#1914
--	-------

Dependencies

bump PyVista and VTK versions (support Python 3.13)	#1924
---	-------

Fixed

docstyle ordering	#1925
adapt Native folder path for Linux and Windows	#1932

Maintenance

update CHANGELOG for v0.10.5	#1919
bump sktionnek/notify-microsoft-teams to v1.0.9 in the actions group	#1920
bump ansys/actions from 9.0.2 to 9.0.3 in the actions group	#1923
fix issues with OSMesa installation and env variables set up	#1927
bump ansys/actions from 9.0.3 to 9.0.6 in the actions group	#1928
pre-commit automatic update	#1930
fix unstable workflows for Linux (missing headless display)	#1931

11.15 0.10.5 - April 16, 2025**Added**

grpc measurement tools stub implementation	#1909
--	-------

Dependencies

bump ansys-api-geometry from 0.4.56 to 0.4.57	#1906
bump grpcio limits and handle erratic gRPC channel creation	#1913

Documentation

Update CONTRIBUTORS.md with the latest contributors	#1907
---	-------

Fixed

is_suppressed is not available until 25R2	#1916
---	-------

Maintenance

update CHANGELOG for v0.10.4	#1901
make doc releases dependent on GH and PyPI release	#1902
bump ansys/actions from 9.0.1 to 9.0.2 in the actions group	#1903
bump skitionek/notify-microsoft-teams from 190d4d92146df11f854709774a4dae6eaf5e2aa3 to fab6aca2609ba706ebc981d066278d47ab4af0fc in the actions group	#1910
pre-commit automatic update	#1911
bump the actions group with 2 updates	#1915
update CHANGELOG for v0.8.3	#1917
update CHANGELOG for v0.9.2	#1918

11.16 0.10.4 - April 09, 2025

Added

grpc named selection stub implementation	#1899
--	-------

Dependencies

bump ansys-api-geometry from 0.4.55 to 0.4.56	#1896
---	-------

Documentation

Ahmed body example for external aero simulation	#1886
adding command for single example build	#1893

Fixed

geomdl dependency in conda-forge	#1900
----------------------------------	-------

Maintenance

update CHANGELOG for v0.10.3	#1894
upgrading to ansys/actions v9 and securing action usage	#1895
bump the actions group with 3 updates	#1897
remove whitelisting	#1898

11.17 0.10.3 - April 08, 2025**Added**

grpc common layer architecture, bodies stub and admin stub implementation	#1867
Logo detection	#1873
DbuApplication stub relocation	#1882

Dependencies

bump ansys-sphinx-theme[autoapi] from 1.3.3 to 1.4.2 in the docs-deps group	#1874
bump ansys-api-geometry from 0.4.50 to 0.4.54	#1875
bump pytest-cov from 6.0.0 to 6.1.0	#1880
bump pytest-cov from 6.1.0 to 6.1.1	#1888
bump ansys-api-geometry from 0.4.54 to 0.4.55	#1889

Documentation

Update CONTRIBUTORS.md with the latest contributors	#1887
---	-------

Fixed

Core Service install location on official build changed	#1876
---	-------

Maintenance

update CHANGELOG for v0.10.2	#1870
pre-commit automatic update	#1878, #1890

Test

issue 1868 (named selection beams testing)	#1871
--	-------

11.18 0.10.2 - March 26, 2025

Added

implement lazy loading of members in NamedSelection to speed up loading times when reading model	#1869
--	-------

Dependencies

bump beartype from 0.19.0 to 0.20.1	#1862
bump beartype from 0.20.1 to 0.20.2	#1864

Maintenance

update CHANGELOG for v0.10.1	#1861
pre-commit automatic update	#1866

Test

issue 1801	#1865
------------	-------

11.19 0.10.1 - March 21, 2025

Maintenance

update CHANGELOG for v0.10.0	#1856
bump version number to 0.11.dev0	#1857
fix release notes inputs	#1858
cleanup deprecated methods	#1860

11.20 0.10.0 - March 21, 2025

Added

named selection functionality	#1768
Streaming upload support	#1779
imprint curves without a sketch	#1781
RGBA color support	#1788
enhanced 3D bounding box implementation	#1805
matrix helper methods	#1806
component name setting	#1820
enable runscript for CoreService	#1821
enhanced beam implementation	#1828
update api geometry dependency	#1834
revolve faces and revolve faces by helix options	#1842
Remove rounds	#1851
blitz (2nd round)	#1853

Dependencies

bump matplotlib from 3.10.0 to 3.10.1	#1789
bump pytest from 8.3.4 to 8.3.5	#1791
bump ansys-api-geometry from 0.4.42 to 0.4.43	#1799
bump ansys-api-geometry from 0.4.43 to 0.4.44	#1803
bump ansys-api-geometry from 0.4.44 to 0.4.45	#1809
bump ansys-api-geometry from 0.4.45 to 0.4.46	#1814
bump pytest-xvfb from 3.0.0 to 3.1.1	#1822
bump ansys-api-geometry from 0.4.46 to 0.4.47	#1827
bump notebook from 7.3.2 to 7.3.3 in the docs-deps group	#1836
bump ansys-api-geometry from 0.4.47 to 0.4.48	#1837
ansys api geometry 0.4.49	#1840
bump numpy from 2.2.3 to 2.2.4	#1844
bump ansys-api-geometry from 0.4.48 to 0.4.49	#1845
bump ansys-api-geometry from 0.4.49 to 0.4.50	#1849

Fixed

flaky color test due to random face assignment	#1794
fix parasolid export tests with more precise backend descriptor	#1802
translating sketch issues when using a custom default unit	#1808
edge start and end were not being mapped correctly	#1816
change Core Service path to executable/DLL after renaming	#1841
tessellation options were not extended to component/face methods	#1850
named selection import test	#1854

Maintenance

update CHANGELOG for v0.9.1	#1787
pre-commit automatic update	#1792, #1810, #1846
remove DMS from pipelines and use core service images only	#1812
use ansys/action/hk-automerge-prs	#1824
upgrading to new features in ansys/actions v8.2	#1852
cleanup blitz PR	#1855

Test

Skip test due to SC bug	#1798
improve share topology test	#1804
Fix slow tests	#1832
adding inward shell	#1833

11.21 0.9.2 - April 16, 2025

11.21.1 Fixed

- is_suppressed is not available until 25R2 #1916

11.22 0.9.1 - 2025-02-28

11.22.1 Added

- offset faces set radius implementation + testing #1769
- separate graphics target #1782

11.22.2 Dependencies

- bump the docs-deps group with 2 updates #1762
- bump ansys-api-geometry from 0.4.38 to 0.4.40 #1763
- bump ansys-sphinx-theme[autoapi] from 1.3.1 to 1.3.2 in the docs-deps group #1766
- bump ansys-tools-visualization-interface from 0.8.1 to 0.8.3 #1767
- bump sphinx from 8.2.0 to 8.2.1 in the docs-deps group #1772
- bump ansys-api-geometry from 0.4.40 to 0.4.42 #1773
- temporary workaround for using trusted publisher approach #1783

11.22.3 Fixed

- allow setting max message length for send operations #1775
- typo in labeler.yml file #1776
- docker build process failing on helper script #1785

11.22.4 Maintenance

- bump dev version to 0.10.dev0 #1752
- update CHANGELOG for v0.9.0 #1760
- pre-commit automatic update #1770

11.23 0.9.0 - 2025-02-18

11.23.1 Added

- design activation changes #1707
- add contributors #1708
- Implementation of inspect & repair geometry #1712
- launch core service from envvar #1716
- workflow enhancements for better tool results #1723
- add face color, round info, bring measure tools to linux #1732
- conservative approach to single design per modeler #1740
- export glb #1741
- allow plotting of individual faces #1757

11.23.2 Dependencies

- bump ansys-api-geometry from 0.4.33 to 0.4.34 #1709
- bump ansys-sphinx-theme[autoapi] from 1.2.6 to 1.2.7 in the docs-deps group #1719
- bump ansys-api-geometry from 0.4.34 to 0.4.35 #1720
- bump ansys-sphinx-theme[autoapi] from 1.2.7 to 1.3.0 in the docs-deps group #1726
- bump ansys-sphinx-theme[autoapi] from 1.3.0 to 1.3.1 in the docs-deps group #1728
- bump ansys-api-geometry from 0.4.35 to 0.4.36 #1729
- bump trame-vtk from 2.8.14 to 2.8.15 #1736
- bump jupyter from 1.16.6 to 1.16.7 in the docs-deps group #1742
- bump ansys-api-geometry from 0.4.36 to 0.4.37 #1743
- bump myst-parser from 4.0.0 to 4.0.1 in the docs-deps group #1744
- bump ansys-api-geometry from 0.4.37 to 0.4.38 #1746
- bump numpy from 2.2.2 to 2.2.3 #1747
- bump panel from 1.6.0 to 1.6.1 #1749
- bump scipy from 1.15.1 to 1.15.2 #1756

11.23.3 Documentation

- update CONTRIBUTING.md #1730

11.23.4 Fixed

- re enable fmd tests #1711
- support body mirror on linux #1714
- use sketch plane for imprint/project curves #1715
- revert boolean ops logic and hold-off on commands-based implementation (temporarily) #1725
- Linux Core Service docker file was missing a dependency #1758

11.23.5 Maintenance

- update CHANGELOG for v0.8.2 #1706
- pre-commit automatic update #1717, #1737
- update SECURITY.md versions supported #1722
- keep simba-plugin-geometry tag #1739
- enhancements to GLB export and object plot() methods #1750
- clean up deprecation warning for trapezoid class and add more info on deprecation #1754

11.23.6 Test

- verifying issue with empty intersect and temporal body creation #1258
- Expand pattern tests #1713
- set body name #1727

- activate 8 linux tests #1745

11.24 0.8.3 - April 16, 2025

11.24.1 Fixed

- is_suppressed is not available until 25R2 #1916

11.25 0.8.2 - 2025-01-29

11.25.1 Added

- create a fillet on an edge/face #1621
- create a full fillet between multiple faces #1623
- extrude existing faces, setup face offset relationships #1628
- interference repair tool #1633
- extrude existing edges to create surface bodies #1638
- create and modify linear patterns #1641
- body suppression state #1643
- parameters refurbished #1647
- rename object #1648
- surface body from trimmed curves #1650
- create circular and fill patterns #1653
- find fix simplify #1661
- replace face #1664
- commands for merge and intersect #1665
- revolve faces a set distance, up to another object, or by a helix #1666
- add split body and tests #1669
- enable get/set persistent ids for stride import/export #1671
- find and fix edge methods #1672
- shell methods #1673
- implementation of NURBS curves #1675
- get assigned material #1684
- matrix rotation and translation #1689
- is_core_service BackendType static method #1692
- export and download stride format #1698
- blitz development #1701

11.25.2 Dependencies

- bump ansys-tools-visualization-interface from 0.7.0 to 0.8.1 #1640
- bump ansys-api-geometry from 0.4.27 to 0.4.28 #1644
- bump sphinx-autodoc-typehints from 3.0.0 to 3.0.1 in the docs-deps group #1651
- bump ansys-api-geometry from 0.4.28 to 0.4.30 #1652
- bump protobuf from 5.28.3 to 5.29.3 in the grpc-deps group across 1 directory #1656
- bump numpy from 2.2.1 to 2.2.2 #1662
- bump ansys-api-geometry from 0.4.30 to 0.4.31 #1663
- bump ansys api geometry from 0.4.30 to 0.4.32 #1677
- bump ansys-api-geometry from 0.4.31 to 0.4.32 #1681
- bump panel from 1.5.5 to 1.6.0 #1682
- bump semver from 3.0.2 to 3.0.4 #1687
- bump ansys-api-geometry from 0.4.32 to 0.4.33 #1695
- bump nbconvert from 7.16.5 to 7.16.6 in the docs-deps group #1700

11.25.3 Fixed

- reactivate test on failing extra edges test #1396
- filter set export id to only CoreService based backends #1685
- cleanup unsupported module #1690
- disable unimplemented tests #1691
- tech review fixes for blitz branch #1703

11.25.4 Maintenance

- update CHANGELOG for v0.8.1 #1639
- whitelist semver package temporarily #1657
- reverting semver package whitelist since problematic version is yanked #1659
- pre-commit automatic update #1667, #1696
- ensure design is closed on test exit #1680
- use dedicate pygeometry-ci-2 runner #1693
- remove towncrier info duplicates #1702

11.25.5 Test

- add more find and fix tests for repair tools #1645
- Add some new tests #1670
- add unit tests for 3 repair tools #1683

11.26 0.8.1 - 2025-01-15

11.26.1 Dependencies

- bump ansys-api-geometry from 0.4.26 to 0.4.27 #1634

11.26.2 Fixed

- release issues encountered #1637

11.26.3 Maintenance

- update CHANGELOG for v0.8.0 #1636

11.27 0.8.0 - 2025-01-15

11.27.1 Added

- active support for Python 3.13 #1481
- add chamfer tool #1495
- allow version input to automatically consider the nuances for the Ansys Student version #1548
- adapt health check timeout algorithm #1559
- add core service support #1571
- enable (partially) prepare and repair tools in Core Service #1580
- create launcher for core services #1587

11.27.2 Dependencies

- bump ansys-api-geometry from 0.4.16 to 0.4.17 #1547
- bump ansys-sphinx-theme[autoapi] from 1.2.1 to 1.2.2 in the docs-deps group #1549
- bump ansys-api-geometry from 0.4.17 to 0.4.18 #1550
- bump ansys-tools-visualization-interface from 0.5.0 to 0.6.0 #1554
- bump pytest from 8.3.3 to 8.3.4 #1562
- bump six from 1.16.0 to 1.17.0 #1568
- bump the docs-deps group across 1 directory with 2 updates #1570
- bump ansys-api-geometry from 0.4.18 to 0.4.20 #1574
- bump numpy from 2.1.3 to 2.2.0 #1575
- bump ansys-api-geometry from 0.4.20 to 0.4.23 #1581
- bump ansys-api-geometry from 0.4.23 to 0.4.24 #1582
- bump ansys-tools-visualization-interface from 0.6.0 to 0.6.1 #1583
- bump ansys-tools-visualization-interface from 0.6.1 to 0.6.2 #1586
- avoid the usage of attrs 24.3.0 (temporary) #1589
- bump jupyter from 1.16.4 to 1.16.5 in the docs-deps group #1590

- bump jupyter from 1.16.5 to 1.16.6 in the docs-deps group #1593
- bump panel from 1.5.4 to 1.5.5 #1595
- bump ansys-sphinx-theme[autoapi] from 1.2.3 to 1.2.4 in the docs-deps group #1597
- bump notebook from 7.3.1 to 7.3.2 in the docs-deps group #1598
- bump numpy from 2.2.0 to 2.2.1 #1599
- bump ansys-tools-path from 0.7.0 to 0.7.1 #1600
- bump nbsphinx from 0.9.5 to 0.9.6 in the docs-deps group #1602
- bump nbconvert from 7.16.4 to 7.16.5 in the docs-deps group #1609
- bump ansys-api-geometry from 0.4.24 to 0.4.25 #1610
- bump sphinx-autodoc-typehints from 2.5.0 to 3.0.0 in the docs-deps group #1611
- bump scipy from 1.14.1 to 1.15.0 #1612
- bump trame-vtk from 2.8.12 to 2.8.13 #1616
- bump trame-vtk from 2.8.13 to 2.8.14 #1617
- bump ansys-tools-visualization-interface from 0.6.2 to 0.7.0 #1619
- bump ansys-sphinx-theme[autoapi] from 1.2.4 to 1.2.6 in the docs-deps group #1624
- bump scipy from 1.15.0 to 1.15.1 #1625
- bump ansys-api-geometry from 0.4.25 to 0.4.26 #1626

11.27.3 Documentation

- Explain how to report a security issue. #1605

11.27.4 Fixed

- numpydoc warnings #1556
- vtk/pyvista issues #1584
- make_child_logger only takes 2 args. #1603
- FAQ on install #1631

11.27.5 Maintenance

- pre-commit automatic update #1366, #1552, #1561, #1588, #1601, #1615, #1630
- update CHANGELOG for v0.7.6 #1545
- change release artifacts self-hosted runner #1546
- automerge pre-commit.ci PRs #1553
- bump pyvista/setup-headless-display-action to v3 #1555
- decouple unstable image promotion #1591
- skip unnecessary stages when containers are the same #1592
- Numpy is already imported at the top of the module. #1604
- update license year using pre-commit hook #1608

11.28 0.7.6 - 2024-11-19

11.28.1 Added

- allow for some additional extrusion direction names #1534

11.28.2 Dependencies

- bump ansys-sphinx-theme[autoapi] from 1.1.7 to 1.2.0 in the docs-deps group #1520
- bump ansys-tools-visualization-interface from 0.4.7 to 0.5.0 #1521
- bump numpy from 2.1.2 to 2.1.3 #1522
- bump ansys-api-geometry from 0.4.13 to 0.4.14 #1525
- bump ansys-api-geometry from 0.4.14 to 0.4.15 #1529
- bump pint from 0.24.3 to 0.24.4 #1530
- bump trame-vtk from 2.8.11 to 2.8.12 #1531
- bump ansys-sphinx-theme[autoapi] from 1.2.0 to 1.2.1 in the docs-deps group #1535
- bump panel from 1.5.3 to 1.5.4 #1536
- bump ansys-tools-path from 0.6.0 to 0.7.0 #1537
- bump ansys-api-geometry from 0.4.15 to 0.4.16 #1538
- limit upper version on grpcio & grpcio-health-checking to 1.68 #1544

11.28.3 Documentation

- typo with the docstrings #1524
- change max header links before more dropdown #1527

11.28.4 Maintenance

- update CHANGELOG for v0.7.5 #1519
- pre-commit automatic update #1523, #1532, #1543
- bump codecov/codecov-action from 4 to 5 in the actions group #1541

11.29 0.7.5 - 2024-10-31

11.29.1 Added

- create body from surface #1454
- performance enhancements to plotter #1496
- allow picking from easy access methods #1499
- implement cut operation in extrude sketch #1510
- caching bodies to avoid unnecessary object creation #1513
- enable retrieval of service logs (via API) #1515

11.29.2 Dependencies

- bump sphinx from 8.1.0 to 8.1.3 in the docs-deps group #1479
- bump ansys-sphinx-theme[autoapi] from 1.1.4 to 1.1.5 in the docs-deps group #1482
- bump the grpc-deps group across 1 directory with 3 updates #1487
- bump ansys-sphinx-theme[autoapi] from 1.1.5 to 1.1.6 in the docs-deps group #1493
- bump trame-vtk from 2.8.10 to 2.8.11 #1494
- bump ansys-api-geometry from 0.4.11 to 0.4.12 #1502
- bump protobuf from 5.28.2 to 5.28.3 in the grpc-deps group #1505
- bump ansys-sphinx-theme[autoapi] from 1.1.6 to 1.1.7 in the docs-deps group #1506
- bump ansys-tools-visualization-interface from 0.4.6 to 0.4.7 #1507
- bump panel from 1.5.2 to 1.5.3 #1508
- bump ansys-api-geometry from 0.4.12 to 0.4.13 #1512
- bump the grpc-deps group with 2 updates #1517
- bump pytest-cov from 5.0.0 to 6.0.0 #1518

11.29.3 Documentation

- avoid having a drop down in the top navigation bar #1485
- provide information on how to build a single example #1490
- add example file to download in the test #1501
- revisit examples to make sure they are properly styled #1509
- align landing page layout with UI/UX requirements #1511

11.29.4 Fixed

- static search options #1478
- respect product_version when launching geometry service #1486

11.29.5 Maintenance

- update CHANGELOG for v0.7.4 #1476
- pre-commit automatic update #1480, #1516
- avoid linkcheck on changelog (unnecessary) #1489
- update CONTRIBUTORS #1492
- allowing new tags for Windows Core Service #1497
- simplify vulnerabilities check #1504

11.30 0.7.4 - 2024-10-11

11.30.1 Dependencies

- bump sphinx from 8.0.2 to 8.1.0 in the docs-deps group #1470

- bump ansys-api-geometry from 0.4.10 to 0.4.11 #1473
- bump ansys-sphinx-theme to v1.1.3 #1475

11.30.2 Fixed

- solving intersphinx warnings on paths #1469
- check_input_types not working with forward refs #1471
- share_topology is available on 24R2 #1472

11.30.3 Maintenance

- update CHANGELOG for v0.7.3 #1466

11.31 0.7.3 - 2024-10-09

11.31.1 Added

- use service colors in plotter (upon request) #1376
- capability to close designs (also on `modeler.exit()`) #1409
- prioritize user-defined SPACECLAIM_MODE env var #1440
- verifying Linux service also accepts colors #1451

11.31.2 Dependencies

- bump protobuf from 5.28.0 to 5.28.1 in the grpc-deps group #1424
- bump the docs-deps group with 2 updates #1425, #1436
- bump ansys-tools-visualization-interface from 0.4.3 to 0.4.4 #1426
- bump pytest from 8.3.2 to 8.3.3 #1427
- bump panel from 1.4.5 to 1.5.0 #1428
- bump protobuf from 5.28.1 to 5.28.2 in the grpc-deps group #1435
- bump the grpc-deps group with 3 updates #1442
- bump beartype from 0.18.5 to 0.19.0 #1443
- bump panel from 1.5.0 to 1.5.1 #1444
- bump ansys-sphinx-theme[autoapi] from 1.1.1 to 1.1.2 in the docs-deps group #1456
- bump ansys-api-geometry from 0.4.8 to 0.4.9 #1457
- bump numpy from 2.1.1 to 2.1.2 #1458
- bump panel from 1.5.1 to 1.5.2 #1459
- bump ansys-api-geometry from 0.4.9 to 0.4.10 #1461
- bump ansys-tools-visualization-interface from 0.4.4 to 0.4.5 #1462
- update protobuf from 5.27.2 to 5.27.5 #1464
- bump sphinx-autodoc-typehints from 2.4.4 to 2.5.0 in the docs-deps group #1465

11.31.3 Documentation

- adding cheat sheet on documentation #1433
- add captions in examples toctrees #1434

11.31.4 Fixed

- ci/cd issues on documentation build #1441
- adapt tessellate tests to new core service #1449
- rename folders on Linux docker image according to new version #1450

11.31.5 Maintenance

- update CHANGELOG for v0.7.2 #1422
- checkout LFS files from previous version to ensure upload #1423
- pre-commit automatic update #1431, #1437, #1445, #1460
- update to ansys actions v8 and docs theme (static search) #1446
- pyvista/setup-headless-display started failing #1447
- check method implemented in Ansys actions #1448
- unstable image promotion and dependabot daily updates #1463

11.32 0.7.2 - 2024-09-11

11.32.1 Added

- allow for platform input when using Ansys Lab #1416
- ensure GrpcClient class closure upon deletion #1417

11.32.2 Dependencies

- bump sphinx-autodoc-typehints from 2.3.0 to 2.4.0 in the docs-deps group #1411
- bump numpy from 2.1.0 to 2.1.1 #1412
- bump ansys-tools-visualization-interface from 0.4.1 to 0.4.3 #1413

11.32.3 Documentation

- remove title from landing page #1408
- adapt examples to use launch_modeler instead of Modeler obj connection #1410

11.32.4 Fixed

- handle properly `np.cross()` - 2d ops deprecated in Numpy 2.X #1419
- change logo link so that it renders properly on PyPI #1420
- wrong path on logo image #1421

11.32.5 Maintenance

- update CHANGELOG for v0.7.1 #1407
- pre-commit automatic update #1418

11.33 0.7.1 - 2024-09-06

11.33.1 Added

- get and set body color #1357
- add `modeler.exit()` method #1375
- setting instance name during component creation #1382
- accept `pathlib.Path` as input in missing methods #1385
- default logs folder on Geometry Service started by Python at PUBLIC (Windows) #1386
- allowing users to specify API version when running script against SpaceClaim or Discovery #1395
- expose `modeler.designs` attribute #1401
- pretty print components #1403

11.33.2 Dependencies

- bump the `grpc-deps` group with 2 updates #1363, #1369
- bump the `docs-deps` group with 2 updates #1364, #1392
- bump `numpy` from 2.0.1 to 2.1.0 #1365
- bump `ansys-sphinx-theme[autoapi]` from 1.0.5 to 1.0.7 in the `docs-deps` group #1370
- bump `ansys-api-geometry` from 0.4.7 to 0.4.8 #1371
- bump `scipy` from 1.14.0 to 1.14.1 #1372
- bump the `grpc-deps` group with 3 updates #1391
- bump `ansys-tools-visualization-interface` from 0.4.0 to 0.4.1 #1393
- bump `ansys-sphinx-theme[autoapi]` from 1.0.7 to 1.0.8 in the `docs-deps` group #1397

11.33.3 Documentation

- add project logo #1405

11.33.4 Fixed

- remove `server_logs_folder` argument for Discovery and SpaceClaim #1387

11.33.5 Maintenance

- update CHANGELOG for v0.7.0 #1360
- bump dev branch to v0.8.dev0 #1361
- solving various warnings #1368
- pre-commit automatic update #1373, #1394
- upload coverage artifacts properly with `upload-artifact@v4.4.0` #1406

11.34 0.7.0 - 2024-08-13

11.34.1 Added

- build: drop support for Python 3.9 #1341
- feat: adapting beartype typehints to +Python 3.10 standard #1347

11.34.2 Dependencies

- build: bump the grpc-deps group with 3 updates #1342
- build: bump panel from 1.4.4 to 1.4.5 #1344
- bump the docs-deps group across 1 directory with 4 updates #1353
- bump trame-vtk from 2.8.9 to 2.8.10 #1355
- bump ansys-api-geometry from 0.4.6 to 0.4.7 #1356

11.34.3 Documentation

- feat: update conf for version 1.x of ansys-sphinx-theme #1351

11.34.4 Fixed

- trapezoid signature change and internal checks #1354

11.34.5 Maintenance

- updating Ansys actions to v7 - changelog related #1348
- ci: bump ansys/actions from 6 to 7 in the actions group #1352
- pre-commit automatic update #1358

11.34.6 Miscellaneous

- chore: pre-commit automatic update #1345

11.35 0.6.6 - 2024-08-01

11.35.1 Added

- feat: Add misc. repair and prepare tool methods #1293
- feat: name setter and fill style getter setters #1299
- feat: extract fluid volume from solid #1306
- feat: keep “other” bodies when performing bool operations #1311
- feat: revolve_sketch rotation definition enhancement #1336

11.35.2 Changed

- chore: update CHANGELOG for v0.6.5 #1290
- chore: enable ruff formatter on pre-commit #1312
- chore: updating dependabot groups #1313

- chore: adding issue links to TODOs #1320
- feat: adapt to new ansys-tools-visualization-interface v0.4.0 #1338

11.35.3 Fixed

- test: create sphere bug raised after box creation #1291
- ci: docker cleanup #1294
- fix: default length units not being used properly on arc creation #1310

11.35.4 Dependencies

- build: bump ansys-api-geometry from 0.4.4 to 0.4.5 #1292
- build: bump pyvista[jupyter] from 0.43.10 to 0.44.0 in the docs-deps group #1296
- build: bump jupyter from 1.16.2 to 1.16.3 in the docs-deps group #1300
- build: bump ansys-api-geometry from 0.4.5 to 0.4.6 #1301
- build: bump pint from 0.24.1 to 0.24.3 #1307
- build: bump grpcio-health-checking from 1.60.0 to 1.64.1 in the grpc-deps group #1315
- build: bump the docs-deps group across 1 directory with 2 updates #1316
- build: bump the grpc-deps group with 2 updates #1322
- build: bump the docs-deps group with 2 updates #1323
- build: bump pyvista[jupyter] from 0.44.0 to 0.44.1 #1324
- build: bump ansys-tools-visualization-interface from 0.2.6 to 0.3.0 #1325
- build: bump pytest from 8.2.2 to 8.3.1 #1326
- build: bump pytest from 8.3.1 to 8.3.2 #1331
- build: bump numpy from 2.0.0 to 2.0.1 #1332

11.35.5 Miscellaneous

- chore: pre-commit automatic update #1327, #1333

11.36 0.6.5 - 2024-07-02

11.36.1 Changed

- chore: update CHANGELOG for v0.6.4 #1278
- build: update sphinx-autodoc-typehints version #1280
- chore: update SECURITY.md #1286

11.36.2 Fixed

- fix: manifest path should render as posix rather than uri #1289

11.36.3 Dependencies

- build: bump protobuf from 5.27.1 to 5.27.2 in the grpc-deps group #1283
- build: bump scipy from 1.13.1 to 1.14.0 #1284
- build: bump vtk from 9.3.0 to 9.3.1 #1287

11.36.4 Miscellaneous

- chore: pre-commit automatic update #1281, #1288

11.37 0.6.4 - 2024-06-24

11.37.1 Added

- feat: using ruff as the main linter/formatter #1274

11.37.2 Changed

- chore: update CHANGELOG for v0.6.3 #1273
- chore: bump pre-commit-hook version #1276

11.37.3 Fixed

- fix: backticks breaking doc build after ruff linter #1275

11.37.4 Dependencies

- build: bump pint from 0.24 to 0.24.1 #1277

11.38 0.6.3 - 2024-06-18

11.38.1 Changed

- chore: update CHANGELOG for v0.6.2 #1263
- build: adapting to numpy 2.x #1265
- docs: using ansys actions (again) to build docs #1270

11.38.2 Fixed

- fix: unnecessary Point3D comparison #1264
- docs: examples are not being uploaded as assets (.py/.ipynb) #1268
- fix: change action order #1269

11.38.3 Dependencies

- build: bump numpy from 1.26.4 to 2.0.0 #1266
- build: bump the docs-deps group with 2 updates #1271

11.38.4 Miscellaneous

- chore: pre-commit automatic update #1267

11.39 0.6.2 - 2024-06-17

11.39.1 Added

- feat: deprecating log_level and logs_folder + adding client log control #1260
- feat: adding deprecation support for args and methods #1261

11.39.2 Changed

- chore: update CHANGELOG for v0.6.1 #1256
- ci: simplify doc build using ansys/actions #1262

11.39.3 Fixed

- fix: Rename built in shadowing variables #1257

11.40 0.6.1 - 2024-06-12

11.40.1 Added

- feat: revolve a sketch given an axis and an origin #1248

11.40.2 Changed

- chore: update CHANGELOG for v0.6.0 #1245
- chore: update dev version to 0.8.dev0 #1246

11.40.3 Fixed

- fix: Bug in *show* function #1255

11.40.4 Dependencies

- build: bump protobuf from 5.27.0 to 5.27.1 in the grpc-deps group #1250
- build: bump the docs-deps group with 2 updates #1251
- build: bump trame-vtk from 2.8.8 to 2.8.9 #1252
- build: bump pint from 0.23 to 0.24 #1253
- build: bump ansys-tools-visualization-interface from 0.2.2 to 0.2.3 #1254

11.40.5 Miscellaneous

- docs: add conda information for package #1247

11.41 0.6.0 - 2024-06-07

11.41.1 Added

- feat: Adapt to ansys-visualizer #959
- fix: rename `GeomPlotter` to `GeometryPlotter` #1227
- refactor: use ansys-tools-visualization-interface global vars rather than env vars #1230
- feat: bump to use v251 as default #1242

11.41.2 Changed

- chore: update CHANGELOG for v0.5.6 #1213
- chore: update SECURITY.md #1214
- ci: use Trusted Publisher for releasing package #1216
- ci: remove pygeometry-ci-1 specific logic #1221
- ci: only run doc build on runners outside the ansys network #1223
- chore: pre-commit automatic update #1224
- ci: announce nightly workflows failing #1237
- ci: failing notifications improvement #1243
- fix: broken interactive docs and improved tests paths #1244

11.41.3 Fixed

- fix: Interactive documentation #1226
- fix: only notify on failure and fill with data #1238

11.41.4 Dependencies

- build: bump protobuf from 5.26.1 to 5.27.0 in the grpc-deps group #1217
- build: bump panel from 1.4.2 to 1.4.3 in the docs-deps group #1218
- build: bump ansys-api-geometry from 0.4.1 to 0.4.2 #1219
- build: bump ansys-sphinx-theme[autoapi] from 0.16.2 to 0.16.5 in the docs-deps group #1231
- build: bump requests from 2.32.2 to 2.32.3 #1232
- build: bump ansys-api-geometry from 0.4.2 to 0.4.3 #1233
- build: bump ansys-tools-visualization-interface from 0.2.1 to 0.2.2 #1234
- build: bump panel from 1.4.3 to 1.4.4 in the docs-deps group #1235
- build: bump ansys-tools-path from 0.5.2 to 0.6.0 #1236
- build: bump grpcio from 1.64.0 to 1.64.1 in the grpc-deps group #1239
- build: bump ansys-api-geometry from 0.4.3 to 0.4.4 #1240
- build: bump pytest from 8.2.1 to 8.2.2 #1241

11.41.5 Miscellaneous

- docs: update AUTHORS #1222

11.42 0.5.6 - 2024-05-23

11.42.1 Added

- feat: add new arc constructors #1208

11.42.2 Changed

- chore: update CHANGELOG for v0.5.5 #1205

11.42.3 Dependencies

- build: bump requests from 2.31.0 to 2.32.2 #1204
- build: bump ansys-sphinx-theme[autoapi] from 0.16.0 to 0.16.2 in the docs-deps group #1210
- build: bump docker from 7.0.0 to 7.1.0 #1211
- build: bump scipy from 1.13.0 to 1.13.1 #1212

11.43 0.5.5 - 2024-05-21

11.43.1 Changed

- docs: adapt ansys_sphinx_theme_autoapi extension for autoapi #1135
- chore: update CHANGELOG for v0.5.4 #1194

11.43.2 Fixed

- fix: adapting Arc class constructor order to (start, end, center) #1196
- chore: limit requests library version under 2.32 #1203

11.43.3 Dependencies

- build: bump grpcio from 1.63.0 to 1.64.0 in the grpc-deps group #1198
- build: bump the docs-deps group with 2 updates #1199
- build: bump pytest from 8.2.0 to 8.2.1 #1200

11.43.4 Miscellaneous

- chore: pre-commit automatic update #1202

11.44 0.5.4 - 2024-05-15

11.44.1 Added

- feat: allow for product_version on geometry service launcher function #1182

11.44.2 Changed

- chore: update CHANGELOG for v0.5.3 #1177

11.44.3 Dependencies

- build: bump the docs-deps group with 4 updates #1178
- build: bump pytest from 8.1.1 to 8.2.0 #1179
- build: bump grpcio from 1.62.2 to 1.63.0 in the grpc-deps group #1186
- build: bump the docs-deps group with 2 updates #1187
- build: bump trame-vtk from 2.8.6 to 2.8.7 #1188
- build: bump nbsphinx from 0.9.3 to 0.9.4 in the docs-deps group #1189
- build: bump trame-vtk from 2.8.7 to 2.8.8 #1190

11.44.4 Miscellaneous

- chore: pre-commit automatic update #1180, #1193
- docs: add geometry preparation for Fluent simulation #1183

11.45 0.5.3 - 2024-04-29

11.45.1 Fixed

- fix: semver intersphinx mapping not resolved properly #1175
- fix: start and end points for edge #1176

11.46 0.5.2 - 2024-04-29

11.46.1 Added

- feat: add semver to intersphinx #1173

11.46.2 Changed

- chore: update CHANGELOG for v0.5.1 #1165
- chore: bump version to v0.6.dev0 #1166
- chore: update CHANGELOG for v0.5.2 #1172
- fix: allow to reuse last release binaries (if requested) #1174

11.46.3 Fixed

- fix: GetSurface and GetCurve not available prior to 24R2 #1171

11.46.4 Miscellaneous

- docs: creating a NACA airfoil example #1167
- docs: simplify README example #1169

11.47 0.5.1 - 2024-04-24

11.47.1 Added

- feat: security updates dropped for v0.3 or earlier #1126
- feat: add `export_to` to functions #1147

11.47.2 Changed

- ci: adapt to vale v3 #1129
- ci: bump ansys/actions from 5 to 6 in the actions group #1133
- docs: add release notes in our documentation #1138
- chore: bump ansys pre-commit hook to v0.3.0 #1139
- chore: use default vale version #1140
- docs: add `user_agent` to Sphinx build #1142
- ci: enabling Linux tests missing #1152
- ci: perform minimal requirements tests #1153

11.47.3 Fixed

- fix: docs link in example #1137
- fix: update backend version message #1145
- fix: Trame issues #1148
- fix: Interactive documentation #1160

11.47.4 Dependencies

- build: bump ansys-tools-path from 0.5.1 to 0.5.2 #1131
- build: bump the grpc-deps group across 1 directory with 3 updates #1156
- build: bump notebook from 7.1.2 to 7.1.3 in the docs-deps group #1157
- build: bump beartype from 0.18.2 to 0.18.5 #1158

11.47.5 Miscellaneous

- docs: add example on exporting designs #1149
- docs: fix link in *CHANGELOG.md* #1154
- chore: pre-commit automatic update #1159

11.48 0.5.0 - 2024-04-17

11.48.1 Added

- feat: inserting document into existing design #930
- feat: add changelog action #1023
- feat: create a sphere body on the backend #1035

- feat: mirror a body #1055
- feat: sweeping chains and profiles #1056
- feat: vulnerability checks #1071
- feat: loft profiles #1075
- feat: accept bandit advisories in-line for subprocess #1077
- feat: adding containers to automatic launcher #1090
- feat: minor changes to Linux Dockerfile #1111
- feat: avoid error if folder exists #1125

11.48.2 Changed

- build: changing sphinx-autoapi from 3.1.a2 to 3.1.a4 #1038
- chore: add pre-commit.ci configuration #1065
- chore: dependabot PR automatic approval #1067
- ci: bump the actions group with 1 update #1082
- chore: update docker tags to be kept #1085
- chore: update pre-commit versions #1094
- build: use ansys-sphinx-theme autoapi target #1097
- fix: removing @PipKat from *.md files - changelog fragments #1098
- ci: dashboard upload does not apply anymore #1099
- chore: pre-commit.ci not working properly #1108
- chore: update and adding pre-commit.ci config hook #1109
- ci: main Python version update to 3.12 #1112
- ci: skip Linux tests with common approach #1113
- ci: build changelog on release #1118
- chore: update CHANGELOG for v0.5.0 #1119

11.48.3 Fixed

- feat: re-enable open file on Linux #817
- fix: adapt export and download tests to new hoops #1057
- fix: linux Dockerfile - replace .NET6.0 references by .NET8.0 #1069
- fix: misleading docstring for sweep_chain() #1070
- fix: prepare_and_start_backend is only available on Windows #1076
- fix: unit tests failing after dms update #1087
- build: beartype upper limit on v0.18 #1095
- fix: improper types being passed for Face and Edge ctor. #1096
- fix: return type should be dict and not ScalarMapContainer (grpc type) #1103
- fix: env version for Dockerfile Windows #1120

- fix: changelog description ill-formatted #1121
- fix: solve issues with intersphinx when releasing #1123

11.48.4 Dependencies

- build: bump the docs-deps group with 2 updates #1062, #1093, #1105
- build: bump ansys-api-geometry from 0.3.13 to 0.4.0 #1066
- build: bump the docs-deps group with 1 update #1080
- build: bump pytest-cov from 4.1.0 to 5.0.0 #1081
- build: bump ansys-api-geometry from 0.4.0 to 0.4.1 #1092
- build: bump beartype from 0.17.2 to 0.18.2 #1106
- build: bump ansys-tools-path from 0.4.1 to 0.5.1 #1107
- build: bump panel from 1.4.0 to 1.4.1 in the docs-deps group #1114
- build: bump scipy from 1.12.0 to 1.13.0 #1115

11.48.5 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`, 35
- `ansys.geometry.core.connection`, 36
 - `backend`, 36
 - `client`, 39
 - `defaults`, 44
 - `docker_instance`, 44
 - `launcher`, 49
 - `product_instance`, 66
 - `validate`, 73
- `ansys.geometry.core.designer`, 74
 - `beam`, 74
 - `body`, 82
 - `component`, 127
 - `coordinate_system`, 148
 - `datumplane`, 150
 - `design`, 152
 - `designpoint`, 162
 - `edge`, 164
 - `face`, 167
 - `geometry_commands`, 175
 - `mating_conditions`, 199
 - `part`, 202
 - `selection`, 204
 - `vertex`, 208
- `ansys.geometry.core.errors`, 435
- `ansys.geometry.core.logger`, 436
- `ansys.geometry.core.materials`, 210
 - `material`, 210
 - `property`, 211
- `ansys.geometry.core.math`, 213
 - `bbox`, 214
 - `constants`, 216
 - `frame`, 217
 - `matrix`, 220
 - `misc`, 224
 - `plane`, 226
 - `point`, 228
 - `vector`, 232
- `ansys.geometry.core.misc`, 240
 - `accuracy`, 240
 - `auxiliary`, 244
 - `checks`, 249
 - `measurements`, 255
 - `options`, 260
 - `units`, 263
- `ansys.geometry.core.modeler`, 446
- `ansys.geometry.core.parameters`, 265
 - `parameter`, 265
- `ansys.geometry.core.plotting`, 268
 - `plotter`, 270
 - `widgets`, 269
 - `show_design_point`, 269
- `ansys.geometry.core.shapes`, 276
 - `box_uv`, 347
 - `curves`, 276
 - `circle`, 276
 - `curve`, 281
 - `curve_evaluation`, 283
 - `ellipse`, 284
 - `line`, 290
 - `nurbs`, 294
 - `trimmed_curve`, 299
 - `parameterization`, 349
 - `surfaces`, 303
 - `cone`, 304
 - `cylinder`, 310
 - `nurbs`, 316
 - `plane`, 323
 - `sphere`, 327

ansys.geometry.core.shapes-surfaces.surface,
333
ansys.geometry.core.shapes-surfaces.surface_evaluation,
335
ansys.geometry.core.shapes-surfaces.torus,
337
ansys.geometry.core.shapes-surfaces.trimmed_surface,
343
ansys.geometry.core.sketch, 358
ansys.geometry.core.sketch.arc, 358
ansys.geometry.core.sketch.box, 361
ansys.geometry.core.sketch.circle, 363
ansys.geometry.core.sketch.edge, 365
ansys.geometry.core.sketch.ellipse, 366
ansys.geometry.core.sketch.face, 368
ansys.geometry.core.sketch.gears, 370
ansys.geometry.core.sketch.nurbs, 373
ansys.geometry.core.sketch.polygon, 375
ansys.geometry.core.sketch.segment, 377
ansys.geometry.core.sketch.sketch, 379
ansys.geometry.core.sketch.slot, 392
ansys.geometry.core.sketch.trapezoid, 393
ansys.geometry.core.sketch.triangle, 395
ansys.geometry.core.tools, 397
ansys.geometry.core.tools.check_geometry, 397
ansys.geometry.core.tools.measurement_tools,
399
ansys.geometry.core.tools.prepare_tools, 401
ansys.geometry.core.tools.problem_areas, 409
ansys.geometry.core.tools.repair_tool_message,
421
ansys.geometry.core.tools.repair_tools, 423
ansys.geometry.core.tools.unsupported, 431
ansys.geometry.core.typing, 452

Symbols

__DOCKER_CLIENT__ (in module *LocalDockInstance*), 47
 __TEMPORARY_BOOL_OPS_FIX__ (in module *ansys.geometry.core.designer.body*), 127
 __add__() (in module *ParamUV*), 351
 __add__() (in module *Point2D*), 229
 __add__() (in module *Point3D*), 231
 __add__() (in module *Vector2D*), 237
 __add__() (in module *Vector3D*), 235
 __call__() (in module *SingletonMeta*), 256
 __eq__() (in module *Arc*), 360
 __eq__() (in module *BoundingBox*), 216
 __eq__() (in module *BoxUV*), 348
 __eq__() (in module *Circle*), 278
 __eq__() (in module *Cone*), 306
 __eq__() (in module *Curve*), 282
 __eq__() (in module *Cylinder*), 313
 __eq__() (in module *Ellipse*), 287
 __eq__() (in module *Frame*), 220
 __eq__() (in module *Interval*), 353
 __eq__() (in module *Line*), 291
 __eq__() (in module *Matrix*), 221
 __eq__() (in module *Measurement*), 258
 __eq__() (in module *NURBSCurve*), 297
 __eq__() (in module *NURBSSurface*), 320
 __eq__() (in module *Plane*), 227
 __eq__() (in module *PlaneSurface*), 325
 __eq__() (in module *Point2D*), 229
 __eq__() (in module *Point3D*), 231
 __eq__() (in module *SketchSegment*), 378
 __eq__() (in module *Sphere*), 330
 __eq__() (in module *Surface*), 334
 __eq__() (in module *Torus*), 339
 __eq__() (in module *Vector2D*), 237
 __eq__() (in module *Vector3D*), 234
 __getitem__() (in module *Logger*), 443
 __iter__() (in module *ParamUV*), 351
 __mod__() (in module *Vector2D*), 237
 __mod__() (in module *Vector3D*), 235
 __mul__() (in module *Matrix*), 221
 __mul__() (in module *ParamUV*), 351
 __mul__() (in module *Vector2D*), 237
 __mul__() (in module *Vector3D*), 235
 __ne__() (in module *Arc*), 360
 __ne__() (in module *BoundingBox*), 216
 __ne__() (in module *BoxUV*), 348
 __ne__() (in module *Frame*), 220
 __ne__() (in module *Matrix*), 221
 __ne__() (in module *Plane*), 227
 __ne__() (in module *Point2D*), 229
 __ne__() (in module *Point3D*), 231
 __ne__() (in module *SketchSegment*), 378
 __ne__() (in module *Vector2D*), 237
 __ne__() (in module *Vector3D*), 234
 __repr__() (in module *Beam*), 81
 __repr__() (in module *BeamCircularProfile*), 76
 __repr__() (in module *BeamCrossSectionInfo*), 78
 __repr__() (in module *Body*), 125
 __repr__() (in module *Component*), 145
 __repr__() (in module *CoordinateSystem*), 150
 __repr__() (in module *DatumPlane*), 151
 __repr__() (in module *Design*), 161
 __repr__() (in module *DesignPoint*), 164
 __repr__() (in module *GrpcClient*), 42
 __repr__() (in module *Interval*), 355
 __repr__() (in module *MasterBody*), 110
 __repr__() (in module *MasterComponent*), 204
 __repr__() (in module *Measurement*), 258
 __repr__() (in module *Modeler*), 450
 __repr__() (in module *NamedSelection*), 208
 __repr__() (in module *ParamUV*), 351
 __repr__() (in module *Parameterization*), 356
 __repr__() (in module *Part*), 203
 __repr__() (in module *TrimmedCurve*), 302
 __repr__() (in module *Vertex*), 209
 __str__() (in module *DesignFileFormat*), 162
 __sub__() (in module *ParamUV*), 351
 __sub__() (in module *Point2D*), 229
 __sub__() (in module *Point3D*), 231
 __sub__() (in module *Vector2D*), 237
 __sub__() (in module *Vector3D*), 235
 __truediv__() (in module *ParamUV*), 351
 __version__ (in module *ansys.geometry.core*), 453

A

ADD (in module *ExtrudeType*), 197

add_beam_circular_profile() (in module *Design*), 158

add_body() (in module *GeometryPlotter*), 273

add_body_edges() (in module *GeometryPlotter*), 272

add_child_logger() (in module *Logger*), 442

add_component() (in module *Component*), 132

add_component() (in module *GeometryPlotter*), 273

add_component_by_body() (in module *GeometryPlotter*), 273

add_design_point() (in module *Component*), 141

add_design_point() (in module *GeometryPlotter*), 274

add_design_points() (in module *Component*), 141

add_face() (in module *GeometryPlotter*), 273

add_frame() (in module *GeometryPlotter*), 271

add_handling_uncaught_exceptions() (in module *Logger*), 443

add_instance_logger() (in module *Logger*), 442

add_material() (in module *Design*), 154

add_members() (in module *NamedSelection*), 207

add_midsurface_offset() (in module *Body*), 115

add_midsurface_offset() (in module *Design*), 160

add_midsurface_offset() (in module *IBody*), 87

add_midsurface_offset() (in module *MasterBody*), 101

add_midsurface_thickness() (in module *Body*), 115

add_midsurface_thickness() (in module *Design*), 159

add_midsurface_thickness() (in module *IBody*), 87

add_midsurface_thickness() (in module *MasterBody*), 100

add_plane() (in module *GeometryPlotter*), 272

add_property() (in module *Material*), 211

add_sketch() (in module *GeometryPlotter*), 272

add_sketch_polydata() (in module *GeometryPlotter*), 274

add_stdout_handler() (in module *ansys.geometry.core.logger*), 445

add_surface() (in module *GeometryPlotter*), 272

addendum (in module *SpurGear*), 372

addfile_handler() (in module *ansys.geometry.core.logger*), 444

ADDINS_SUBFOLDER (in module *ansys.geometry.core.connection.product_instance*), 71

ANCHOR_LOCATION (in module *SectionAnchorType*), 82

angle (in module *Arc*), 359

ANGLE (in module *DefaultUnitsClass*), 257

angle (in module *Polygon*), 376

angle (in module *SketchEllipse*), 368

ANGLE_ACCURACY (in module *ansys.geometry.core.misc.accuracy*), 244

angle_accuracy() (in module *Accuracy*), 241

angle_deviation (in module *TessellationOptions*), 263

angle_is_negative() (in module *Accuracy*), 243

angle_is_positive() (in module *Accuracy*), 243

angle_is_zero() (in module *Accuracy*), 242

ansys.geometry.core

module, 35

ansys.geometry.core.connection

module, 36

ansys.geometry.core.connection.backend

module, 36

ansys.geometry.core.connection.backend.ApiVersions

(built-in class), 38

ansys.geometry.core.connection.backend.BackendType

(built-in class), 36

ansys.geometry.core.connection.client

module, 39

ansys.geometry.core.connection.client.GrpcClient

(built-in class), 39

ansys.geometry.core.connection.defaults

module, 44

ansys.geometry.core.connection.docker_instance

module, 44

ansys.geometry.core.connection.docker_instance.GeometryCor

(built-in class), 47

ansys.geometry.core.connection.docker_instance.LocalDocker

(built-in class), 45

ansys.geometry.core.connection.launcher

module, 49

ansys.geometry.core.connection.product_instance

module, 66

ansys.geometry.core.connection.product_instance.ProductIns

(built-in class), 67

ansys.geometry.core.connection.validate

module, 73

ansys.geometry.core.designer

module, 74

ansys.geometry.core.designer.beam

module, 74

ansys.geometry.core.designer.beam.Beam (built-in class), 79

ansys.geometry.core.designer.beam.BeamCircularProfile

(built-in class), 75

ansys.geometry.core.designer.beam.BeamCrossSectionInfo

(built-in class), 76

ansys.geometry.core.designer.beam.BeamProfile

(built-in class), 74

ansys.geometry.core.designer.beam.BeamProperties

(built-in class), 78

ansys.geometry.core.designer.beam.BeamType

(built-in class), 81

ansys.geometry.core.designer.beam.SectionAnchorType

(built-in class), 82

ansys.geometry.core.designer.body

module, 82
 ansys.geometry.core.designer.body.Body (built-in class), 111
 ansys.geometry.core.designer.body.CollisionType (built-in class), 125
 ansys.geometry.core.designer.body.FillStyle (built-in class), 126
 ansys.geometry.core.designer.body.IBody (built-in class), 83
 ansys.geometry.core.designer.body.MasterBody (built-in class), 96
 ansys.geometry.core.designer.body.MidSurfaceOffsetType (built-in class), 125
 ansys.geometry.core.designer.component module, 127
 ansys.geometry.core.designer.component.Component (built-in class), 128
 ansys.geometry.core.designer.component.Extrusion (built-in class), 147
 ansys.geometry.core.designer.component.SharedTopology (built-in class), 147
 ansys.geometry.core.designer.component.SweepWithSideData (built-in class), 127
 ansys.geometry.core.designer.coordinate_system module, 148
 ansys.geometry.core.designer.coordinate_system.CoordinateSystem (built-in class), 148
 ansys.geometry.core.designer.datumplane module, 150
 ansys.geometry.core.designer.datumplane.DatumPlane (built-in class), 150
 ansys.geometry.core.designer.design module, 152
 ansys.geometry.core.designer.design.Design (built-in class), 152
 ansys.geometry.core.designer.design.DesignFileFormat (built-in class), 161
 ansys.geometry.core.designer.designpoint module, 162
 ansys.geometry.core.designer.designpoint.DesignPoint (built-in class), 162
 ansys.geometry.core.designer.edge module, 164
 ansys.geometry.core.designer.edge.CurveType (built-in class), 166
 ansys.geometry.core.designer.edge.Edge (built-in class), 164
 ansys.geometry.core.designer.face module, 167
 ansys.geometry.core.designer.face.Face (built-in class), 168
 ansys.geometry.core.designer.face.FaceLoop (built-in class), 167
 ansys.geometry.core.designer.face.FaceLoopType (built-in class), 174
 ansys.geometry.core.designer.face.SurfaceType (built-in class), 174
 ansys.geometry.core.designer.geometry_commands module, 175
 ansys.geometry.core.designer.geometry_commands.DraftSide (built-in class), 199
 ansys.geometry.core.designer.geometry_commands.ExtrudeType (built-in class), 197
 ansys.geometry.core.designer.geometry_commands.FillPattern (built-in class), 198
 ansys.geometry.core.designer.geometry_commands.GeometryCommand (built-in class), 175
 ansys.geometry.core.designer.geometry_commands.OffsetMode (built-in class), 198
 ansys.geometry.core.designer.mating_conditions module, 199
 ansys.geometry.core.designer.mating_conditions.AlignCondition (built-in class), 200
 ansys.geometry.core.designer.mating_conditions.MatingCondition (built-in class), 199
 ansys.geometry.core.designer.mating_conditions.OrientCondition (built-in class), 201
 ansys.geometry.core.designer.mating_conditions.TangentCondition (built-in class), 201
 ansys.geometry.core.designer.part module, 202
 ansys.geometry.core.designer.part.MasterComponent (built-in class), 203
 ansys.geometry.core.designer.part.Part (built-in class), 202
 ansys.geometry.core.designer.selection module, 204
 ansys.geometry.core.designer.selection.NamedSelection (built-in class), 205
 ansys.geometry.core.designer.vertex module, 208
 ansys.geometry.core.designer.vertex.Vertex (built-in class), 208
 ansys.geometry.core.errors module, 435
 ansys.geometry.core.errors.GeometryExitedError, 436
 ansys.geometry.core.errors.GeometryRuntimeError, 435
 ansys.geometry.core.logger module, 436
 ansys.geometry.core.logger.InstanceFilter (built-in class), 439
 ansys.geometry.core.logger.Logger (built-in class), 440
 ansys.geometry.core.logger.PyGeometryCustomAdapter (built-in class), 437
 ansys.geometry.core.logger.PyGeometryFormatter

(built-in class), 439
 ansys.geometry.core.logger.PyGeometryPercentStyle (built-in class), 439
 ansys.geometry.core.materials
 module, 210
 ansys.geometry.core.materials.material
 module, 210
 ansys.geometry.core.materials.material.Material (built-in class), 210
 ansys.geometry.core.materials.property
 module, 211
 ansys.geometry.core.materials.property.MaterialProperty (built-in class), 211
 ansys.geometry.core.materials.property.MaterialPropertyType (built-in class), 212
 ansys.geometry.core.math
 module, 213
 ansys.geometry.core.math.bbox
 module, 214
 ansys.geometry.core.math.bbox.BoundingBox (built-in class), 214
 ansys.geometry.core.math.constants
 module, 216
 ansys.geometry.core.math.frame
 module, 217
 ansys.geometry.core.math.frame.Frame (built-in class), 217
 ansys.geometry.core.math.matrix
 module, 220
 ansys.geometry.core.math.matrix.Matrix (built-in class), 220
 ansys.geometry.core.math.matrix.Matrix33 (built-in class), 221
 ansys.geometry.core.math.matrix.Matrix44 (built-in class), 221
 ansys.geometry.core.math.misc
 module, 224
 ansys.geometry.core.math.plane
 module, 226
 ansys.geometry.core.math.plane.Plane (built-in class), 226
 ansys.geometry.core.math.point
 module, 228
 ansys.geometry.core.math.point.Point2D (built-in class), 228
 ansys.geometry.core.math.point.Point3D (built-in class), 229
 ansys.geometry.core.math.vector
 module, 232
 ansys.geometry.core.math.vector.UnitVector2D (built-in class), 239
 ansys.geometry.core.math.vector.UnitVector3D (built-in class), 238
 ansys.geometry.core.math.vector.Vector2D (built-in class), 235
 ansys.geometry.core.math.vector.Vector3D (built-in class), 232
 ansys.geometry.core.misc
 module, 240
 ansys.geometry.core.misc.accuracy
 module, 240
 ansys.geometry.core.misc.accuracy.Accuracy (built-in class), 240
 ansys.geometry.core.misc.auxiliary
 module, 244
 ansys.geometry.core.misc.checks
 module, 246
 ansys.geometry.core.misc.measurements
 module, 255
 ansys.geometry.core.misc.measurements.Angle (built-in class), 259
 ansys.geometry.core.misc.measurements.Area (built-in class), 259
 ansys.geometry.core.misc.measurements.DefaultUnitsClass (built-in class), 256
 ansys.geometry.core.misc.measurements.Distance (built-in class), 258
 ansys.geometry.core.misc.measurements.Measurement (built-in class), 257
 ansys.geometry.core.misc.measurements.SingletonMeta (built-in class), 256
 ansys.geometry.core.misc.options
 module, 260
 ansys.geometry.core.misc.options.ImportOptions (built-in class), 260
 ansys.geometry.core.misc.options.ImportOptionsDefinitions (built-in class), 261
 ansys.geometry.core.misc.options.TessellationOptions (built-in class), 262
 ansys.geometry.core.misc.units
 module, 263
 ansys.geometry.core.misc.units.PhysicalQuantity (built-in class), 264
 ansys.geometry.core.modeler
 module, 446
 ansys.geometry.core.modeler.Modeler (built-in class), 446
 ansys.geometry.core.parameters
 module, 265
 ansys.geometry.core.parameters.parameter
 module, 265
 ansys.geometry.core.parameters.parameter.Parameter (built-in class), 265
 ansys.geometry.core.parameters.parameter.ParameterType (built-in class), 267
 ansys.geometry.core.parameters.parameter.ParameterUpdateStrategy (built-in class), 268

ansys.geometry.core.plotting
 module, 268
 ansys.geometry.core.plotting.plotter
 module, 270
 ansys.geometry.core.plotting.plotter.GeometryPlotter
 (built-in class), 270
 ansys.geometry.core.plotting.widgets
 module, 269
 ansys.geometry.core.plotting.widgets.show_design_parameters
 module, 269
 ansys.geometry.core.plotting.widgets.show_design_parameters
 (built-in class), 269
 ansys.geometry.core.shapes
 module, 276
 ansys.geometry.core.shapes.box_uv
 module, 347
 ansys.geometry.core.shapes.box_uv.BoxUV
 (built-in class), 347
 ansys.geometry.core.shapes.box_uv.LocationUV
 (built-in class), 348
 ansys.geometry.core.shapes.curves
 module, 276
 ansys.geometry.core.shapes.curves.circle
 module, 276
 ansys.geometry.core.shapes.curves.circle.Circle
 (built-in class), 276
 ansys.geometry.core.shapes.curves.circle.CircleArcSegment
 (built-in class), 280
 ansys.geometry.core.shapes.curves.curve
 module, 281
 ansys.geometry.core.shapes.curves.curve.Curve
 (built-in class), 282
 ansys.geometry.core.shapes.curves.curve_evaluation
 module, 283
 ansys.geometry.core.shapes.curves.curve_evaluation
 (built-in class), 283
 ansys.geometry.core.shapes.curves.ellipse
 module, 284
 ansys.geometry.core.shapes.curves.ellipse.Ellipse
 (built-in class), 285
 ansys.geometry.core.shapes.curves.ellipse.EllipseArcSegment
 (built-in class), 288
 ansys.geometry.core.shapes.curves.line
 module, 290
 ansys.geometry.core.shapes.curves.line.Line
 (built-in class), 290
 ansys.geometry.core.shapes.curves.line.LineEvaluation
 (built-in class), 293
 ansys.geometry.core.shapes.curves.nurbs
 module, 294
 ansys.geometry.core.shapes.curves.nurbs.NURBSCurve
 (built-in class), 294
 ansys.geometry.core.shapes.curves.nurbs.NURBSCurveEvaluation
 (built-in class), 298
 ansys.geometry.core.shapes.curves.trimmed_curve
 module, 299
 ansys.geometry.core.shapes.curves.trimmed_curve.ReversedTrimmedCurve
 (built-in class), 302
 ansys.geometry.core.shapes.curves.trimmed_curve.TrimmedCurve
 (built-in class), 299
 ansys.geometry.core.shapes.parameterization
 module, 349
 ansys.geometry.core.shapes.parameterization.Interval
 (built-in class), 352
 ansys.geometry.core.shapes.parameterization.Parameterization
 (built-in class), 355
 ansys.geometry.core.shapes.parameterization.ParamForm
 (built-in class), 356
 ansys.geometry.core.shapes.parameterization.ParamType
 (built-in class), 357
 ansys.geometry.core.shapes.parameterization.ParamUV
 (built-in class), 350
 ansys.geometry.core.shapes-surfaces
 module, 303
 ansys.geometry.core.shapes-surfaces.cone
 module, 304
 ansys.geometry.core.shapes-surfaces.cone.Cone
 (built-in class), 304
 ansys.geometry.core.shapes-surfaces.cone.ConeEvaluation
 (built-in class), 307
 ansys.geometry.core.shapes-surfaces.cylinder
 module, 310
 ansys.geometry.core.shapes-surfaces.cylinder.Cylinder
 (built-in class), 310
 ansys.geometry.core.shapes-surfaces.cylinder.CylinderEvaluation
 (built-in class), 314
 ansys.geometry.core.shapes-surfaces.nurbs
 module, 316
 ansys.geometry.core.shapes-surfaces.nurbs.NURBSurface
 (built-in class), 316
 ansys.geometry.core.shapes-surfaces.nurbs.NURBSurfaceEvaluation
 (built-in class), 321
 ansys.geometry.core.shapes-surfaces.plane
 module, 323
 ansys.geometry.core.shapes-surfaces.plane.PlaneEvaluation
 (built-in class), 326
 ansys.geometry.core.shapes-surfaces.plane.PlaneSurface
 (built-in class), 323
 ansys.geometry.core.shapes-surfaces.sphere
 module, 327
 ansys.geometry.core.shapes-surfaces.sphere.Sphere
 (built-in class), 328
 ansys.geometry.core.shapes-surfaces.sphere.SphereEvaluation
 (built-in class), 331
 ansys.geometry.core.shapes-surfaces.surface
 module, 333
 ansys.geometry.core.shapes-surfaces.surface.Surface
 (built-in class), 333

ansys.geometry.core.shapes-surfaces.surface_evaluation module, 335	ansys.geometry.core.sketch.polygon module, 375
ansys.geometry.core.shapes-surfaces.surface_evaluation (built-in class), 335	ansys.geometry.core.sketch.polygon.Polygon (built-in class), 375
ansys.geometry.core.shapes-surfaces.torus module, 337	ansys.geometry.core.sketch.segment module, 377
ansys.geometry.core.shapes-surfaces.torus.Torus (built-in class), 337	ansys.geometry.core.sketch.segment.SketchSegment (built-in class), 377
ansys.geometry.core.shapes-surfaces.torus.TorusEvaluation module, 340	ansys.geometry.core.sketch.sketch module, 379
ansys.geometry.core.shapes-surfaces.trimmed_surfaces module, 343	ansys.geometry.core.sketch.sketch.Sketch (built-in class), 379
ansys.geometry.core.shapes-surfaces.trimmed_surfaces (built-in class), 343	ansys.geometry.core.sketch.sketch.slot module, 392
ansys.geometry.core.shapes-surfaces.trimmed_surfaces (built-in class), 343	ansys.geometry.core.sketch.slot.Slot (built-in class), 392
ansys.geometry.core.sketch module, 358	ansys.geometry.core.sketch.trapezoid module, 393
ansys.geometry.core.sketch.arc module, 358	ansys.geometry.core.sketch.trapezoid.Trapezoid (built-in class), 394
ansys.geometry.core.sketch.arc.Arc (built-in class), 358	ansys.geometry.core.sketch.triangle module, 395
ansys.geometry.core.sketch.box module, 361	ansys.geometry.core.sketch.triangle.Triangle (built-in class), 395
ansys.geometry.core.sketch.box.Box (built-in class), 362	ansys.geometry.core.tools module, 397
ansys.geometry.core.sketch.circle module, 363	ansys.geometry.core.tools.check_geometry module, 397
ansys.geometry.core.sketch.circle.SketchCircle (built-in class), 363	ansys.geometry.core.tools.check_geometry.GeometryIssue (built-in class), 397
ansys.geometry.core.sketch.edge module, 365	ansys.geometry.core.tools.check_geometry.InspectResult (built-in class), 398
ansys.geometry.core.sketch.edge.SketchEdge (built-in class), 365	ansys.geometry.core.tools.measurement_tools module, 399
ansys.geometry.core.sketch.ellipse module, 366	ansys.geometry.core.tools.measurement_tools.Gap (built-in class), 399
ansys.geometry.core.sketch.ellipse.SketchEllipse (built-in class), 366	ansys.geometry.core.tools.measurement_tools.MeasurementTools (built-in class), 399
ansys.geometry.core.sketch.face module, 368	ansys.geometry.core.tools.prepare_tools module, 401
ansys.geometry.core.sketch.face.SketchFace (built-in class), 369	ansys.geometry.core.tools.prepare_tools.EnclosureOptions (built-in class), 401
ansys.geometry.core.sketch.gears module, 370	ansys.geometry.core.tools.prepare_tools.PrepareTools (built-in class), 402
ansys.geometry.core.sketch.gears.DummyGear (built-in class), 370	ansys.geometry.core.tools.problem_areas module, 409
ansys.geometry.core.sketch.gears.Gear (built-in class), 370	ansys.geometry.core.tools.problem_areas.DuplicateFaceProblem (built-in class), 410
ansys.geometry.core.sketch.gears.SpurGear (built-in class), 371	ansys.geometry.core.tools.problem_areas.ExtraEdgeProblemArea (built-in class), 413
ansys.geometry.core.sketch.nurbs module, 373	ansys.geometry.core.tools.problem_areas.InexactEdgeProblemArea (built-in class), 412
ansys.geometry.core.sketch.nurbs.SketchNurbs (built-in class), 373	ansys.geometry.core.tools.problem_areas.InterferenceProblemArea (built-in class), 419

ansys.geometry.core.tools.problem_areas.LogoProblemAreas (built-in class), 420
 ansys.geometry.core.tools.problem_areas.MissingFaceProblemAreas (built-in class), 411
 ansys.geometry.core.tools.problem_areas.ProblemArea (built-in class), 409
 ansys.geometry.core.tools.problem_areas.ShortEdgeProblemAreas (built-in class), 414
 ansys.geometry.core.tools.problem_areas.SmallFaceProblemAreas (built-in class), 415
 ansys.geometry.core.tools.problem_areas.SplitEdgeProblemAreas (built-in class), 416
 ansys.geometry.core.tools.problem_areas.StitchFaceProblemAreas (built-in class), 417
 ansys.geometry.core.tools.problem_areas.UnsimplifiedFaceProblemAreas (built-in class), 418
 ansys.geometry.core.tools.repair_tool_message module, 421
 ansys.geometry.core.tools.repair_tool_message.RepairToolMessage (built-in class), 421
 ansys.geometry.core.tools.repair_tools module, 423
 ansys.geometry.core.tools.repair_tools.RepairTools (built-in class), 423
 ansys.geometry.core.tools.unsupported module, 431
 ansys.geometry.core.tools.unsupported.ExportIdData (built-in class), 431
 ansys.geometry.core.tools.unsupported.PersistentIdType (built-in class), 434
 ansys.geometry.core.tools.unsupported.UnsupportedCommands (built-in class), 432
 ansys.geometry.core.typing module, 452
 ANSYS_GEOMETRY_SERVICE_ROOT (in module ansys.geometry.core.connection.product_instance), 73
 apex (in module Cone), 306
 apex_param (in module Cone), 306
 arc() (in module Sketch), 383
 arc_from_start_center_and_angle() (in module Sketch), 385
 arc_from_start_end_and_radius() (in module Sketch), 385
 arc_from_three_points() (in module Sketch), 384
 arc_to_point() (in module Sketch), 384
 area (in module BeamProperties), 79
 area (in module Box), 363
 area (in module Circle), 278
 AREA (in module DefaultUnitsClass), 257
 area (in module Ellipse), 287
 area (in module Face), 170
 area (in module Polygon), 376
 area (in module Slot), 393
 assign_material() (in module Body), 114
 assign_material() (in module IBody), 87
 assign_problem_areas() (in module MasterBody), 100
 BACK (in module DraftSide), 199
 BACKEND_API_VERSION_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_API_VERSION_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_DISCOVERY_HIDDEN (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_HOST_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_LOG_LEVEL_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_LOGS_FOLDER_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_PORT_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_SPACECLAIM_HIDDEN (in module ansys.geometry.core.connection.product_instance), 73
 BACKEND_SPACECLAIM_HIDDEN_ENVVAR_KEY (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_SPACECLAIM_HIDDEN_ENVVAR_VALUE (in module ansys.geometry.core.connection.product_instance), 73
 BACKEND_SPACECLAIM_OPTIONS (in module ansys.geometry.core.connection.product_instance), 72
 BACKEND_SPLASH_OFF (in module ansys.geometry.core.connection.product_instance), 73
 BACKEND_SUBFOLDER (in module ansys.geometry.core.connection.product_instance), 71
 BACKEND_TRACE_VARIABLE (in module ansys.geometry.core.connection.product_instance), 72
 backend_type (in module GrpcClient), 42
 backend_version (in module GrpcClient), 42
 base_diameter (in module SpurGear), 372

base_unit (in module *PhysicalQuantity*), 264
 base_unit() (in module *Point2D*), 229
 base_unit() (in module *Point3D*), 231
 BASE_UNIT_LENGTH (in module *ansys.geometry.core.math.point*), 232
 base_width (in module *Trapezoid*), 395
 BEAM (in module *BeamType*), 81
 beam_profiles (in module *Design*), 154
 beams (in module *Component*), 131
 beams (in module *NamedSelection*), 206
 bodies (in module *Component*), 131
 bodies (in module *InterferenceProblemAreas*), 420
 bodies (in module *NamedSelection*), 206
 bodies (in module *Part*), 203
 bodies (in module *StitchFaceProblemAreas*), 418
 body (in module *Edge*), 165
 body (in module *Face*), 170
 body (in module *InspectResult*), 398
 body (in module *Vertex*), 209
 BOTTOM (in module *MidSurfaceOffsetType*), 125
 BottomCenter (in module *LocationUV*), 349
 BottomLeft (in module *LocationUV*), 349
 BottomRight (in module *LocationUV*), 349
 bounding_box (in module *Body*), 114
 bounding_box (in module *Edge*), 166
 bounding_box (in module *Face*), 170
 bounding_box (in module *MasterBody*), 99
 bounding_box() (in module *IBody*), 86
 box() (in module *Sketch*), 388
 box_uv (in module *TrimmedSurface*), 344
 build_repair_tool_message() (in module *ProblemArea*), 410

C

CABLE (in module *BeamType*), 81
 cache (in module *TorusEvaluation*), 343
 callback() (in module *ShowDesignPoints*), 270
 center (in module *Arc*), 359
 center (in module *BeamCircularProfile*), 76
 center (in module *BoundingBox*), 215
 center (in module *Box*), 362
 Center (in module *LocationUV*), 349
 center (in module *Polygon*), 376
 center (in module *SketchCircle*), 364
 center (in module *SketchEllipse*), 368
 center (in module *Slot*), 393
 center (in module *Trapezoid*), 395
 centroid (in module *BeamProperties*), 79
 CENTROID (in module *SectionAnchorType*), 82
 chamfer() (in module *GeometryCommands*), 177
 channel (in module *GrpcClient*), 42
 check_is_float_int() (in module *ansys.geometry.core.misc.checks*), 250

check_ndarray_is_all_nan() (in module *ansys.geometry.core.misc.checks*), 251
 check_ndarray_is_float_int() (in module *ansys.geometry.core.misc.checks*), 250
 check_ndarray_is_non_zero() (in module *ansys.geometry.core.misc.checks*), 251
 check_ndarray_is_not_none() (in module *ansys.geometry.core.misc.checks*), 251
 check_nurbs_compatibility() (in module *ansys.geometry.core.misc.checks*), 253
 check_pint_unit_compatibility() (in module *ansys.geometry.core.misc.checks*), 252
 check_type() (in module *ansys.geometry.core.misc.checks*), 252
 check_type_all_elements_in_iterable() (in module *ansys.geometry.core.misc.checks*), 252
 check_type_equivalence() (in module *ansys.geometry.core.misc.checks*), 252
 circle (in module *CircleEvaluation*), 280
 circle() (in module *Sketch*), 387
 CIRCULAR (in module *ParamType*), 357
 cleanup_bodies (in module *ImportOptions*), 261
 client (in module *Modeler*), 448
 close() (in module *Design*), 154
 close() (in module *GrpcClient*), 42
 close() (in module *Modeler*), 449
 close() (in module *ProductInstance*), 68
 CLOSED (in module *ParamForm*), 357
 color (in module *Body*), 113
 color (in module *Face*), 170
 color (in module *MasterBody*), 98
 color() (in module *IBody*), 85
 combine_merge() (in module *Body*), 124
 combine_merge() (in module *IBody*), 96
 combine_merge() (in module *MasterBody*), 108
 compare_with_tolerance() (in module *Accuracy*), 241
 components (in module *Component*), 131
 components (in module *NamedSelection*), 206
 components (in module *Part*), 203
 cone (in module *ConeEvaluation*), 308
 CONSTRAINED_PARAMETERS (in module *ParameterUpdateStatus*), 268
 CONTAINED (in module *CollisionType*), 126
 CONTAINEDTOUCH (in module *CollisionType*), 126
 container (in module *LocalDockInstance*), 46
 contains() (in module *BoxUV*), 348
 contains() (in module *Interval*), 355
 contains_param() (in module *Circle*), 280
 contains_param() (in module *Cone*), 307
 contains_param() (in module *Curve*), 282
 contains_param() (in module *Cylinder*), 313
 contains_param() (in module *Ellipse*), 288
 contains_param() (in module *Line*), 293

- `contains_param()` (in module *NURBSCurve*), 297
- `contains_param()` (in module *NURBSSurface*), 320
- `contains_param()` (in module *PlaneSurface*), 325
- `contains_param()` (in module *Sphere*), 331
- `contains_param()` (in module *Surface*), 334
- `contains_param()` (in module *Torus*), 340
- `contains_point()` (in module *BoundingBox*), 215
- `contains_point()` (in module *Circle*), 280
- `contains_point()` (in module *Cone*), 307
- `contains_point()` (in module *Curve*), 282
- `contains_point()` (in module *Cylinder*), 314
- `contains_point()` (in module *Ellipse*), 288
- `contains_point()` (in module *Line*), 293
- `contains_point()` (in module *NURBSCurve*), 297
- `contains_point()` (in module *NURBSSurface*), 321
- `contains_point()` (in module *PlaneSurface*), 325
- `contains_point()` (in module *SketchEdge*), 366
- `contains_point()` (in module *SketchNurbs*), 374
- `contains_point()` (in module *Sphere*), 331
- `contains_point()` (in module *Surface*), 334
- `contains_point()` (in module *Torus*), 340
- `contains_point_components()` (in module *BoundingBox*), 215
- `contains_value()` (in module *Interval*), 355
- `control_points` (in module *NURBSCurve*), 295
- `control_points` (in module *NURBSSurface*), 318
- `control_points` (in module *SketchNurbs*), 374
- `convert_color_to_hex()` (in module *ansys.geometry.core.misc.auxiliary*), 248
- `convert_opacity_to_hex()` (in module *ansys.geometry.core.misc.auxiliary*), 249
- `convert_quantity_to_server_units()` (in module *Parameter*), 266
- `coordinate_systems` (in module *Component*), 131
- `copy()` (in module *Body*), 119
- `copy()` (in module *IBody*), 90
- `copy()` (in module *MasterBody*), 105
- `copy_faces()` (in module *Component*), 143
- `CORE_GEOMETRY_SERVICE_EXE` (in module *ansys.geometry.core.connection.product_instance*), 72
- `CORE_GEOMETRY_SERVICE_FOLDER` (in module *ansys.geometry.core.connection.product_instance*), 71
- `CORE_LINUX` (in module *BackendType*), 37
- `CORE_LINUX_25_2` (in module *GeometryContainers*), 48
- `CORE_LINUX_26_1` (in module *GeometryContainers*), 48
- `CORE_LINUX_27_1` (in module *GeometryContainers*), 49
- `CORE_LINUX_LATEST` (in module *GeometryContainers*), 48
- `CORE_LINUX_LATEST_UNSTABLE` (in module *GeometryContainers*), 48
- `CORE_WINDOWS` (in module *BackendType*), 37
- `CORE_WINDOWS_25_2` (in module *GeometryContainers*), 48
- `CORE_WINDOWS_26_1` (in module *GeometryContainers*), 48
- `CORE_WINDOWS_27_1` (in module *GeometryContainers*), 49
- `CORE_WINDOWS_LATEST` (in module *GeometryContainers*), 48
- `CORE_WINDOWS_LATEST_UNSTABLE` (in module *GeometryContainers*), 48
- `create_align_condition()` (in module *GeometryCommands*), 191
- `create_beam()` (in module *Component*), 140
- `create_beams()` (in module *Component*), 140
- `create_body_from_loft_profile()` (in module *Component*), 137
- `create_body_from_loft_profiles_with_guides()` (in module *Component*), 137
- `create_body_from_surface()` (in module *Component*), 139
- `create_box_enclosure()` (in module *PrepareTools*), 406
- `create_circular_pattern()` (in module *GeometryCommands*), 183
- `create_coordinate_system()` (in module *Component*), 139
- `create_cylinder_enclosure()` (in module *PrepareTools*), 407
- `create_datum_plane()` (in module *Component*), 141
- `create_design()` (in module *Modeler*), 448
- `create_fill_pattern()` (in module *GeometryCommands*), 185
- `create_isoparametric_curves()` (in module *Face*), 172
- `create_linear_pattern()` (in module *GeometryCommands*), 182
- `create_matrix_from_mapping()` (in module *Matrix44*), 224
- `create_matrix_from_rotation_about_axis()` (in module *Matrix44*), 223
- `create_named_selection()` (in module *Design*), 157
- `create_orient_condition()` (in module *GeometryCommands*), 193
- `create_repair_message_from_response()` (in module *ansys.geometry.core.tools.repair_tool_message*), 423
- `create_rotation()` (in module *Matrix44*), 223
- `create_shared_topology` (in module *EnclosureOptions*), 401
- `create_sphere()` (in module *Component*), 136
- `create_sphere_enclosure()` (in module *PrepareTools*), 408
- `create_surface()` (in module *Component*), 138

- create_surface_from_face() (in module Component), 138
- create_surface_from_trimmed_curves() (in module Component), 139
- create_tangent_condition() (in module GeometryCommands), 192
- create_translation() (in module Matrix44), 222
- created_bodies (in module RepairToolMessage), 422
- CRITICAL (in module ansys.geometry.core.logger), 445
- critical (in module Logger), 441
- cross() (in module Vector2D), 237
- cross() (in module Vector3D), 234
- curvature (in module CircleEvaluation), 281
- curvature (in module CurveEvaluation), 284
- curvature (in module EllipseEvaluation), 290
- curvature (in module LineEvaluation), 294
- curvature (in module NURBSCurveEvaluation), 299
- curvature (in module TorusEvaluation), 342
- curve_type (in module Edge), 165
- CURVETYPE_CIRCLE (in module CurveType), 167
- CURVETYPE_ELLIPSE (in module CurveType), 167
- CURVETYPE_LINE (in module CurveType), 167
- CURVETYPE_NURBS (in module CurveType), 167
- CURVETYPE_PROCEDURAL (in module CurveType), 167
- CURVETYPE_UNKNOWN (in module CurveType), 167
- cushion_proportion (in module EnclosureOptions), 401
- CUSTOM (in module MidSurfaceOffsetType), 125
- CUT (in module ExtrudeType), 197
- cylinder (in module CylinderEvaluation), 314
- ## D
- datum_planes (in module Component), 131
- DEBUG (in module ansys.geometry.core.logger), 445
- debug (in module Logger), 441
- dedendum (in module SpurGear), 372
- DEFAULT (in module FillStyle), 126
- DEFAULT_COLOR (in module ansys.geometry.core.misc.auxiliary), 249
- DEFAULT_FILE_HEADER (in module ansys.geometry.core.logger), 445
- DEFAULT_HOST (in module ansys.geometry.core.connection.defaults), 44
- DEFAULT_MATRIX33 (in module ansys.geometry.core.math.matrix), 224
- DEFAULT_MATRIX44 (in module ansys.geometry.core.math.matrix), 224
- DEFAULT_PIM_CONFIG (in module ansys.geometry.core.connection.defaults), 44
- DEFAULT_POINT2D (in module ansys.geometry.core.math.constants), 217
- DEFAULT_POINT2D_VALUES (in module ansys.geometry.core.math.point), 232
- DEFAULT_POINT3D (in module ansys.geometry.core.math.constants), 217
- DEFAULT_POINT3D_VALUES (in module ansys.geometry.core.math.point), 232
- DEFAULT_PORT (in module ansys.geometry.core.connection.defaults), 44
- DEFAULT_STDOUT_HEADER (in module ansys.geometry.core.logger), 445
- DEFAULT_UNITS (in module ansys.geometry.core.misc.measurements), 259
- degree (in module NURBSCurve), 295
- degree (in module SketchNurbs), 374
- degree_u (in module NURBSSurface), 318
- degree_v (in module NURBSSurface), 318
- delete_beam() (in module Component), 142
- delete_beam_profile() (in module Design), 160
- delete_body() (in module Component), 141
- delete_component() (in module Component), 141
- delete_component() (in module Design), 158
- delete_datum_plane() (in module Component), 142
- delete_named_selection() (in module Design), 158
- deleted_bodies (in module RepairToolMessage), 422
- DENSITY (in module MaterialPropertyType), 213
- deprecated_argument() (in module ansys.geometry.core.misc.checks), 254
- deprecated_method() (in module ansys.geometry.core.misc.checks), 253
- design (in module Modeler), 448
- DESIGN_FILE_FORMAT_EXTENSIONS (in module ansys.geometry.core.designer.design), 162
- design_id (in module Design), 154
- design_points (in module Component), 131
- design_points (in module NamedSelection), 206
- detach() (in module Face), 173
- detach_faces() (in module Body), 124
- detach_faces() (in module GeometryCommands), 196
- detect_helices() (in module PrepareTools), 405
- detect_sweepable_bodies() (in module PrepareTools), 408
- determinant() (in module Matrix), 221
- diameter (in module Circle), 278
- dimension_type (in module Parameter), 266
- dimension_value (in module Parameter), 266
- DIMENSIONTYPE_ANGULAR (in module ParameterType), 267
- DIMENSIONTYPE_ARC (in module ParameterType), 267
- DIMENSIONTYPE_AREA (in module ParameterType), 267
- DIMENSIONTYPE_COUNT (in module ParameterType), 267
- DIMENSIONTYPE_DIAMETRIC (in module ParameterType), 267
- DIMENSIONTYPE_LINEAR (in module ParameterType), 267
- DIMENSIONTYPE_MASS (in module ParameterType), 267

DIMENSIONTYPE_RADIAL (in module *ParameterType*), 267

DIMENSIONTYPE_UNITLESS (in module *ParameterType*), 267

DIMENSIONTYPE_UNKNOWN (in module *ParameterType*), 267

DIMENSIONTYPE_VOLUME (in module *ParameterType*), 267

dir_x (in module *Circle*), 278

dir_x (in module *Cone*), 305

dir_x (in module *Cylinder*), 311

dir_x (in module *Ellipse*), 286

dir_x (in module *NURBSSurface*), 318

dir_x (in module *PlaneSurface*), 325

dir_x (in module *Sphere*), 329

dir_x (in module *Torus*), 339

dir_y (in module *Circle*), 278

dir_y (in module *Cone*), 305

dir_y (in module *Cylinder*), 311

dir_y (in module *Ellipse*), 286

dir_y (in module *PlaneSurface*), 325

dir_y (in module *Sphere*), 329

dir_y (in module *Torus*), 339

dir_z (in module *Circle*), 278

dir_z (in module *Cone*), 305

dir_z (in module *Cylinder*), 311

dir_z (in module *Ellipse*), 286

dir_z (in module *NURBSSurface*), 318

dir_z (in module *PlaneSurface*), 325

dir_z (in module *Sphere*), 329

dir_z (in module *Torus*), 339

direction (in module *Line*), 291

direction_x (in module *BeamCircularProfile*), 76

direction_x (in module *Frame*), 219

direction_y (in module *BeamCircularProfile*), 76

direction_y (in module *Frame*), 219

direction_z (in module *Frame*), 219

DISABLE_ACTIVE_DESIGN_CHECK (in module *ansys.geometry.core*), 453

DISABLE_MULTIPLE_DESIGN_CHECK (in module *ansys.geometry.core*), 453

DISCO (in module *DesignFileFormat*), 162

DISCOVERY (in module *BackendType*), 37

DISCOVERY_EXE (in module *ansys.geometry.core.connection.product_instance*), 72

DISCOVERY_FOLDER (in module *ansys.geometry.core.connection.product_instance*), 71

DISCOVERY_HEADLESS (in module *BackendType*), 37

distance (in module *Gap*), 399

docker_client() (in module *LocalDockerInstance*), 47

DOCUMENTATION_BUILD (in module *ansys.geometry.core*), 453

DOUBLE_ACCURACY (in module *ansys.geometry.core.misc.accuracy*), 244

double_accuracy() (in module *Accuracy*), 241

download() (in module *Design*), 154

draft_faces() (in module *GeometryCommands*), 194

dummy_gear() (in module *Sketch*), 390

E

eccentricity (in module *Ellipse*), 286

edge() (in module *Sketch*), 382

edges (in module *Body*), 113

edges (in module *ExtraEdgeProblemAreas*), 414

edges (in module *Face*), 170

edges (in module *FaceLoop*), 168

edges (in module *GeometryIssue*), 398

edges (in module *InexactEdgeProblemAreas*), 413

edges (in module *MasterBody*), 99

edges (in module *MissingFaceProblemAreas*), 412

edges (in module *NamedSelection*), 206

edges (in module *ShortEdgeProblemAreas*), 415

edges (in module *Sketch*), 380

edges (in module *SketchFace*), 369

edges (in module *SplitEdgeProblemAreas*), 417

edges() (in module *IBody*), 85

ELASTIC_MODULUS (in module *MaterialPropertyType*), 213

ellipse (in module *EllipseEvaluation*), 289

ellipse() (in module *Sketch*), 389

end (in module *Arc*), 359

end (in module *Beam*), 80

end (in module *Edge*), 166

end (in module *Interval*), 353

end (in module *SketchEdge*), 365

end (in module *SketchNurbs*), 374

end (in module *SketchSegment*), 378

end (in module *TrimmedCurve*), 300

enhanced_share_topology() (in module *PrepareTools*), 404

ensure_design_is_active() (in module *ansys.geometry.core.misc.checks*), 250

equal_doubles() (in module *Accuracy*), 241

ERROR (in module *ansys.geometry.core.logger*), 445

error (in module *Logger*), 441

ERROR_GRAPHICS_REQUIRED (in module *ansys.geometry.core.misc.checks*), 255

evaluate() (in module *Circle*), 278

evaluate() (in module *Cone*), 306

evaluate() (in module *Curve*), 282

evaluate() (in module *Cylinder*), 313

evaluate() (in module *DatumPlane*), 151

evaluate() (in module *Ellipse*), 287

evaluate() (in module *Line*), 291

evaluate() (in module *NURBSCurve*), 297

evaluate() (in module *NURBSSurface*), 320

- `evaluate()` (in module *PlaneSurface*), 326
 - `evaluate()` (in module *Sphere*), 330
 - `evaluate()` (in module *Surface*), 334
 - `evaluate()` (in module *Torus*), 339
 - `evaluate_proportion()` (in module *ReversedTrimmedCurve*), 303
 - `evaluate_proportion()` (in module *TrimmedCurve*), 301
 - `evaluate_proportion()` (in module *TrimmedSurface*), 345
 - `existed_previously` (in module *LocalDockerInstance*), 46
 - `exit()` (in module *Modeler*), 449
 - `export_glb()` (in module *GeometryPlotter*), 275
 - `export_to_disco()` (in module *Design*), 155
 - `export_to_fmd()` (in module *Design*), 156
 - `export_to_iges()` (in module *Design*), 156
 - `export_to_parasolid_bin()` (in module *Design*), 156
 - `export_to_parasolid_text()` (in module *Design*), 156
 - `export_to_pmdb()` (in module *Design*), 157
 - `export_to_scdocx()` (in module *Design*), 155
 - `export_to_step()` (in module *Design*), 156
 - `export_to_stride()` (in module *Design*), 155
 - `extract_volume_from_edge_loops()` (in module *PrepareTools*), 403
 - `extract_volume_from_faces()` (in module *PrepareTools*), 402
 - `extrude_edges()` (in module *GeometryCommands*), 180
 - `extrude_edges_up_to()` (in module *GeometryCommands*), 181
 - `extrude_face()` (in module *Component*), 136
 - `extrude_faces()` (in module *GeometryCommands*), 178
 - `extrude_faces_up_to()` (in module *GeometryCommands*), 179
 - `extrude_sketch()` (in module *Component*), 133
- ## F
- `face()` (in module *Sketch*), 381
 - `face_ids` (in module *LogoProblemArea*), 421
 - `faces` (in module *Body*), 113
 - `faces` (in module *DuplicateFaceProblemAreas*), 411
 - `faces` (in module *Edge*), 165
 - `faces` (in module *GeometryIssue*), 398
 - `faces` (in module *MasterBody*), 99
 - `faces` (in module *NamedSelection*), 206
 - `faces` (in module *Sketch*), 380
 - `faces` (in module *SmallFaceProblemAreas*), 416
 - `faces` (in module *UnsimplifiedFaceProblemAreas*), 419
 - `faces()` (in module *IBody*), 85
 - `FAILURE` (in module *ParameterUpdateStatus*), 268
 - `file_handler` (in module *Logger*), 441
 - `file_handler` (in module *PyGeometryCustomAdapter*), 438
 - `FILE_MSG_FORMAT` (in module *ansys.geometry.core.logger*), 445
 - `FILE_NAME` (in module *ansys.geometry.core.logger*), 445
 - `fill_style` (in module *Body*), 112
 - `fill_style` (in module *MasterBody*), 98
 - `fill_style()` (in module *IBody*), 84
 - `fillet()` (in module *GeometryCommands*), 177
 - `filter()` (in module *InstanceFilter*), 439
 - `find_and_fix_extra_edges()` (in module *RepairTools*), 428
 - `find_and_fix_short_edges()` (in module *RepairTools*), 428
 - `find_and_fix_simplify()` (in module *RepairTools*), 429
 - `find_and_fix_split_edges()` (in module *RepairTools*), 429
 - `find_and_fix_stitch_faces()` (in module *RepairTools*), 430
 - `find_and_remove_logos()` (in module *PrepareTools*), 405
 - `find_duplicate_faces()` (in module *RepairTools*), 425
 - `find_extra_edges()` (in module *RepairTools*), 425
 - `find_inexact_edges()` (in module *RepairTools*), 425
 - `find_interferences()` (in module *RepairTools*), 427
 - `find_logos()` (in module *PrepareTools*), 404
 - `find_missing_faces()` (in module *RepairTools*), 426
 - `find_short_edges()` (in module *RepairTools*), 425
 - `find_simplify()` (in module *RepairTools*), 427
 - `find_small_faces()` (in module *RepairTools*), 426
 - `find_split_edges()` (in module *RepairTools*), 424
 - `find_stitch_faces()` (in module *RepairTools*), 427
 - `first_derivative` (in module *CircleEvaluation*), 281
 - `first_derivative` (in module *CurveEvaluation*), 284
 - `first_derivative` (in module *EllipseEvaluation*), 289
 - `first_derivative` (in module *LineEvaluation*), 294
 - `first_derivative` (in module *NURBSCurveEvaluation*), 298
 - `fit_curve_from_points()` (in module *NURBSCurve*), 296
 - `fit_curve_from_points()` (in module *SketchNurbs*), 375
 - `fit_surface_from_points()` (in module *NURBSSurface*), 319
 - `fix()` (in module *DuplicateFaceProblemAreas*), 411
 - `fix()` (in module *ExtraEdgeProblemAreas*), 414
 - `fix()` (in module *InexactEdgeProblemAreas*), 413
 - `fix()` (in module *InterferenceProblemAreas*), 420
 - `fix()` (in module *LogoProblemArea*), 421
 - `fix()` (in module *MissingFaceProblemAreas*), 412
 - `fix()` (in module *ProblemArea*), 410
 - `fix()` (in module *ShortEdgeProblemAreas*), 415

- fix() (in module *SmallFaceProblemAreas*), 416
 fix() (in module *SplitEdgeProblemAreas*), 417
 fix() (in module *StitchFaceProblemAreas*), 418
 fix() (in module *UnsimplifiedFaceProblemAreas*), 419
 flat (in module *Point2D*), 229
 flat (in module *Point3D*), 231
 FMD (in module *DesignFileFormat*), 162
 FORCE_ADD (in module *ExtrudeType*), 197
 FORCE_CUT (in module *ExtrudeType*), 197
 FORCE_INDEPENDENT (in module *ExtrudeType*), 197
 FORCE_NEW_SURFACE (in module *ExtrudeType*), 197
 form (in module *Parameterization*), 356
 found (in module *RepairToolMessage*), 422
 frame (in module *CoordinateSystem*), 149
 frame (in module *EnclosureOptions*), 401
 from_control_points() (in module *NURBSCurve*), 296
 from_control_points() (in module *NURBSSurface*), 319
 from_id() (in module *MaterialPropertyType*), 213
 from_points() (in module *UnitVector2D*), 239
 from_points() (in module *UnitVector3D*), 238
 from_points() (in module *Vector2D*), 237
 from_points() (in module *Vector3D*), 235
 from_start_center_and_angle() (in module *Arc*), 361
 from_start_end_and_radius() (in module *Arc*), 360
 from_string() (in module *ExtrusionDirection*), 148
 from_three_points() (in module *Arc*), 360
 full_fillet() (in module *GeometryCommands*), 178
- ## G
- geomdl_nurbs_curve (in module *NURBSCurve*), 295
 geomdl_nurbs_curve (in module *SketchNurbs*), 374
 geomdl_nurbs_surface (in module *NURBSSurface*), 318
 geometry (in module *TrimmedCurve*), 300
 geometry (in module *TrimmedSurface*), 344
 geometry_commands (in module *Modeler*), 448
 GEOMETRY_SERVICE_DOCKER_IMAGE (in module *ansys.geometry.core.connection.defaults*), 44
 GEOMETRY_SERVICE_EXE (in module *ansys.geometry.core.connection.product_instance*), 71
 get() (in module *Sketch*), 381
 get_active_design() (in module *Modeler*), 448
 get_all_bodies() (in module *Component*), 132
 get_all_bodies_from_design() (in module *ansys.geometry.core.misc.auxiliary*), 245
 get_all_parameters() (in module *Design*), 159
 get_angle_between() (in module *Vector2D*), 237
 get_angle_between() (in module *Vector3D*), 234
 get_assigned_material() (in module *Body*), 115
 get_assigned_material() (in module *IBody*), 87
 get_assigned_material() (in module *MasterBody*), 100
 get_available_port() (in module *ansys.geometry.core.connection.product_instance*), 71
 get_beams_from_ids() (in module *ansys.geometry.core.misc.auxiliary*), 248
 get_bodies_from_ids() (in module *ansys.geometry.core.misc.auxiliary*), 246
 get_body_occurrences_from_import_id() (in module *UnsupportedCommands*), 433
 get_bounding_box() (in module *Body*), 114
 get_bounding_box() (in module *Edge*), 166
 get_bounding_box() (in module *Face*), 170
 get_bounding_box() (in module *IBody*), 86
 get_bounding_box() (in module *MasterBody*), 100
 get_center() (in module *BoxUV*), 348
 get_collision() (in module *Body*), 119
 get_collision() (in module *IBody*), 90
 get_collision() (in module *MasterBody*), 105
 get_components_from_ids() (in module *ansys.geometry.core.misc.auxiliary*), 246
 get_corner() (in module *BoxUV*), 348
 get_design_from_body() (in module *ansys.geometry.core.misc.auxiliary*), 245
 get_design_from_component() (in module *ansys.geometry.core.misc.auxiliary*), 244
 get_design_from_edge() (in module *ansys.geometry.core.misc.auxiliary*), 245
 get_design_from_face() (in module *ansys.geometry.core.misc.auxiliary*), 245
 get_design_points_from_ids() (in module *ansys.geometry.core.misc.auxiliary*), 248
 get_edge_occurrences_from_import_id() (in module *UnsupportedCommands*), 434
 get_edges_from_ids() (in module *ansys.geometry.core.misc.auxiliary*), 247
 get_face_occurrences_from_import_id() (in module *UnsupportedCommands*), 434
 get_faces_from_ids() (in module *ansys.geometry.core.misc.auxiliary*), 246
 get_geometry_container_type() (in module *ansys.geometry.core.connection.docker_instance*), 49
 get_multiplier() (in module *ExtrusionDirection*), 148
 get_name() (in module *GrpcClient*), 42
 get_named_selections() (in module *Beam*), 81
 get_named_selections() (in module *Body*), 119
 get_named_selections() (in module *Component*), 146
 get_named_selections() (in module *DesignPoint*), 164
 get_named_selections() (in module *Edge*), 166
 get_named_selections() (in module *Face*), 172

[get_named_selections\(\) \(in module IBody\), 91](#)
[get_named_selections\(\) \(in module MasterBody\), 105](#)
[get_named_selections\(\) \(in module Vertex\), 209](#)
[get_proportional_parameters\(\) \(in module TrimmedSurface\), 344](#)
[get_raw_tessellation\(\) \(in module Body\), 119](#)
[get_raw_tessellation\(\) \(in module Design\), 160](#)
[get_raw_tessellation\(\) \(in module IBody\), 91](#)
[get_raw_tessellation\(\) \(in module MasterBody\), 105](#)
[get_relative_val\(\) \(in module Interval\), 353](#)
[get_round_info\(\) \(in module GeometryCommands\), 189](#)
[get_service_logs\(\) \(in module Modeler\), 451](#)
[get_span\(\) \(in module Interval\), 353](#)
[get_two_circle_intersections\(\) \(in module ansys.geometry.core.math.misc\), 225](#)
[get_vertices_from_ids\(\) \(in module ansys.geometry.core.misc.auxiliary\), 247](#)
[get_vtk_tessellation\(\) \(in module Body\), 121](#)
[get_vtk_tessellation\(\) \(in module IBody\), 92](#)
[get_vtk_tessellation\(\) \(in module MasterBody\), 107](#)
[get_world_transform\(\) \(in module Component\), 132](#)
[global_to_local_rotation \(in module Frame\), 219](#)
[graphics_required\(\) \(in module ansys.geometry.core.misc.checks\), 254](#)
[GRID \(in module FillPatternType\), 198](#)
[guide \(in module SweepWithGuideData\), 128](#)

H

[half_angle \(in module Cone\), 305](#)
[handler\(\) \(in module ansys.geometry.core.errors\), 436](#)
[healthy \(in module GrpcClient\), 42](#)
[height \(in module Box\), 362](#)
[height \(in module Cone\), 305](#)
[height \(in module Slot\), 393](#)
[height \(in module Trapezoid\), 395](#)

I

[id \(in module Beam\), 80](#)
[id \(in module BeamProfile\), 75](#)
[id \(in module Body\), 112](#)
[id \(in module Component\), 131](#)
[id \(in module CoordinateSystem\), 149](#)
[id \(in module DatumPlane\), 151](#)
[id \(in module DesignPoint\), 163](#)
[id \(in module Edge\), 165](#)
[id \(in module Face\), 169](#)
[id \(in module MasterBody\), 98](#)
[id \(in module MasterComponent\), 204](#)
[id \(in module MatingCondition\), 200](#)
[id \(in module NamedSelection\), 206](#)

[id \(in module Parameter\), 266](#)
[id \(in module Part\), 203](#)
[id \(in module ProblemArea\), 410](#)
[id \(in module Vertex\), 209](#)
[id\(\) \(in module IBody\), 84](#)
[id_type \(in module ExportIdData\), 432](#)
[IDENTITY_MATRIX33 \(in module ansys.geometry.core.math.constants\), 217](#)
[IDENTITY_MATRIX44 \(in module ansys.geometry.core.math.constants\), 217](#)
[IGES \(in module DesignFileFormat\), 162](#)
[IGNORE_RELATIONSHIPS \(in module OffsetMode\), 198](#)
[import_coordinate_systems \(in module ImportOptions\), 261](#)
[import_curves \(in module ImportOptions\), 261](#)
[import_hidden_components_and_geometry \(in module ImportOptions\), 261](#)
[import_named_selections \(in module ImportOptions\), 261](#)
[import_named_selections\(\) \(in module Component\), 146](#)
[import_named_selections_keys \(in module ImportOptionsDefinitions\), 262](#)
[import_names \(in module ImportOptions\), 261](#)
[import_planes \(in module ImportOptions\), 261](#)
[import_points \(in module ImportOptions\), 261](#)
[imprint_curves\(\) \(in module Body\), 115](#)
[imprint_curves\(\) \(in module IBody\), 87](#)
[imprint_curves\(\) \(in module MasterBody\), 101](#)
[imprint_projected_curves\(\) \(in module Body\), 116](#)
[imprint_projected_curves\(\) \(in module IBody\), 88](#)
[imprint_projected_curves\(\) \(in module MasterBody\), 102](#)
[inflate\(\) \(in module BoxUV\), 348](#)
[inflate\(\) \(in module Interval\), 355](#)
[INFO \(in module ansys.geometry.core.logger\), 445](#)
[info \(in module Logger\), 441](#)
[INNER_LOOP \(in module FaceLoopType\), 175](#)
[inner_radius \(in module Polygon\), 376](#)
[insert_file\(\) \(in module Design\), 160](#)
[inspect_geometry\(\) \(in module RepairTools\), 430](#)
[instance_name \(in module Component\), 131](#)
[INTERSECT \(in module CollisionType\), 126](#)
[intersect\(\) \(in module Body\), 123](#)
[intersect\(\) \(in module IBody\), 95](#)
[intersect\(\) \(in module Interval\), 354](#)
[intersect\(\) \(in module MasterBody\), 109](#)
[intersect_bboxes\(\) \(in module BoundingBox\), 216](#)
[intersect_curve\(\) \(in module TrimmedCurve\), 301](#)
[intersect_curve_and_surface\(\) \(in module GeometryCommands\), 196](#)
[intersect_interval\(\) \(in module ansys.geometry.core.math.misc\), 225](#)
[interval \(in module Parameterization\), 356](#)

- `interval` (in module *TrimmedCurve*), 300
 - `interval_u` (in module *BoxUV*), 348
 - `interval_v` (in module *BoxUV*), 348
 - `INVALID` (in module *DesignFileFormat*), 162
 - `inverse()` (in module *Matrix*), 221
 - `is_active` (in module *Design*), 154
 - `is_alive` (in module *Beam*), 80
 - `is_alive` (in module *Body*), 113
 - `is_alive` (in module *Component*), 131
 - `is_alive` (in module *CoordinateSystem*), 149
 - `is_alive` (in module *DatumPlane*), 151
 - `is_alive` (in module *MasterBody*), 99
 - `is_alive()` (in module *IBody*), 85
 - `is_clockwise` (in module *Arc*), 359
 - `is_closed` (in module *Design*), 154
 - `is_closed` (in module *GrpcClient*), 42
 - `is_closed()` (in module *Interval*), 353
 - `is_coincident_circle()` (in module *Circle*), 279
 - `is_coincident_ellipse()` (in module *Ellipse*), 287
 - `is_coincident_line()` (in module *Line*), 292
 - `is_core_service()` (in module *BackendType*), 37
 - `is_deleted` (in module *MatingCondition*), 200
 - `is_docker_installed()` (in module *LocalDockerInstance*), 47
 - `is_empty()` (in module *BoxUV*), 348
 - `is_empty()` (in module *Interval*), 353
 - `is_enabled` (in module *MatingCondition*), 200
 - `is_headless_service()` (in module *BackendType*), 37
 - `is_linux_service()` (in module *BackendType*), 37
 - `is_negative()` (in module *BoxUV*), 348
 - `is_negative()` (in module *Interval*), 354
 - `is_open()` (in module *Interval*), 353
 - `is_opposite()` (in module *Vector2D*), 237
 - `is_opposite()` (in module *Vector3D*), 234
 - `is_opposite_line()` (in module *Line*), 292
 - `is_parallel_to()` (in module *Vector2D*), 237
 - `is_parallel_to()` (in module *Vector3D*), 234
 - `is_perpendicular_to()` (in module *Vector2D*), 237
 - `is_perpendicular_to()` (in module *Vector3D*), 234
 - `is_point_contained()` (in module *Plane*), 227
 - `is_reversed` (in module *AlignCondition*), 200
 - `is_reversed` (in module *Edge*), 165
 - `is_reversed` (in module *Face*), 169
 - `is_reversed` (in module *OrientCondition*), 201
 - `is_reversed` (in module *TangentCondition*), 201
 - `is_satisfied` (in module *MatingCondition*), 200
 - `is_set()` (in module *CurveEvaluation*), 284
 - `is_suppressed` (in module *Body*), 113
 - `is_suppressed` (in module *MasterBody*), 98
 - `is_suppressed()` (in module *IBody*), 84
 - `is_surface` (in module *Body*), 113
 - `is_surface` (in module *MasterBody*), 98
 - `is_surface()` (in module *IBody*), 86
 - `is_translation()` (in module *Matrix44*), 222
 - `is_valid` (in module *AlignCondition*), 200
 - `is_valid` (in module *OrientCondition*), 201
 - `is_valid` (in module *TangentCondition*), 201
 - `is_within_tolerance()` (in module *Accuracy*), 243
 - `is_zero` (in module *Vector2D*), 236
 - `is_zero` (in module *Vector3D*), 233
 - `issues` (in module *InspectResult*), 398
 - `ixx` (in module *BeamProperties*), 79
 - `ixy` (in module *BeamProperties*), 79
 - `iyy` (in module *BeamProperties*), 79
- ## K
- `knots` (in module *NURBSCurve*), 296
 - `knots` (in module *SketchNurbs*), 374
 - `knotvector_u` (in module *NURBSSurface*), 318
 - `knotvector_v` (in module *NURBSSurface*), 318
 - `kwargs_passed_not_accepted()` (in module *ansys.geometry.core.misc.checks*), 254
- ## L
- `launch_docker_modeler()` (in module *ansys.geometry.core.connection.launcher*), 51
 - `launch_modeler()` (in module *ansys.geometry.core.connection.launcher*), 50
 - `launch_modeler_with_core_service()` (in module *ansys.geometry.core.connection.launcher*), 63
 - `launch_modeler_with_discovery()` (in module *ansys.geometry.core.connection.launcher*), 58
 - `launch_modeler_with_discovery_and_pimlight()` (in module *ansys.geometry.core.connection.launcher*), 52
 - `launch_modeler_with_geometry_service()` (in module *ansys.geometry.core.connection.launcher*), 55
 - `launch_modeler_with_geometry_service_and_pimlight()` (in module *ansys.geometry.core.connection.launcher*), 53
 - `launch_modeler_with_spaceclaim()` (in module *ansys.geometry.core.connection.launcher*), 61
 - `launch_modeler_with_spaceclaim_and_pimlight()` (in module *ansys.geometry.core.connection.launcher*), 54
 - `launch_remote_modeler()` (in module *ansys.geometry.core.connection.launcher*), 50
 - `LeftCenter` (in module *LocationUV*), 349
 - `length` (in module *Arc*), 359
 - `LENGTH` (in module *DefaultUnitsClass*), 257

length (in module *Edge*), 165
length (in module *FaceLoop*), 168
length (in module *Polygon*), 376
length (in module *SketchEdge*), 365
length (in module *SketchSegment*), 378
length (in module *TrimmedCurve*), 300
length() (in module *NURBSCurve*), 296
LENGTH_ACCURACY (in module *ansys.geometry.core.misc.accuracy*), 244
length_accuracy() (in module *Accuracy*), 241
length_is_equal() (in module *Accuracy*), 241
length_is_greater_than_or_equal() (in module *Accuracy*), 242
length_is_less_than_or_equal() (in module *Accuracy*), 242
length_is_negative() (in module *Accuracy*), 242
length_is_positive() (in module *Accuracy*), 242
length_is_zero() (in module *Accuracy*), 242
level (in module *Logger*), 441
level (in module *PyGeometryCustomAdapter*), 438
line (in module *LineEvaluation*), 293
LINEAR (in module *ParamType*), 357
linear_eccentricity (in module *Ellipse*), 287
LINK_TRUSS (in module *BeamType*), 81
LINUX_SERVICE (in module *BackendType*), 37
local_to_global_rotation (in module *Frame*), 219
LOG (in module *ansys.geometry.core.logger*), 446
log (in module *GrpcClient*), 42
log (in module *Logger*), 441
LOG_LEVEL (in module *ansys.geometry.core.logger*), 445
log_to_file() (in module *Logger*), 441
log_to_file() (in module *PyGeometryCustomAdapter*), 438
log_to_stdout() (in module *Logger*), 442
log_to_stdout() (in module *PyGeometryCustomAdapter*), 438
logger (in module *Logger*), 441
logger (in module *PyGeometryCustomAdapter*), 438
loops (in module *Face*), 170

M

magnitude (in module *Vector2D*), 236
magnitude (in module *Vector3D*), 233
major_radius (in module *Ellipse*), 286
major_radius (in module *Torus*), 339
make_independent() (in module *Component*), 146
MANIFEST_FILENAME (in module *ansys.geometry.core.connection.product_instance*), 71
map() (in module *Body*), 118
map() (in module *IBody*), 90
map() (in module *MasterBody*), 104
material (in module *Body*), 114
material (in module *MasterBody*), 99

material() (in module *IBody*), 86
materials (in module *Design*), 154
max_aspect_ratio (in module *TessellationOptions*), 263
max_bbox (in module *FaceLoop*), 168
max_corner (in module *BoundingBox*), 215
max_curvature (in module *ConeEvaluation*), 309
max_curvature (in module *CylinderEvaluation*), 316
max_curvature (in module *NURBSSurfaceEvaluation*), 323
max_curvature (in module *PlaneEvaluation*), 327
max_curvature (in module *SphereEvaluation*), 333
max_curvature (in module *SurfaceEvaluation*), 337
max_curvature (in module *TorusEvaluation*), 343
max_curvature_direction (in module *ConeEvaluation*), 309
max_curvature_direction (in module *CylinderEvaluation*), 316
max_curvature_direction (in module *NURBSSurfaceEvaluation*), 323
max_curvature_direction (in module *PlaneEvaluation*), 327
max_curvature_direction (in module *SphereEvaluation*), 333
max_curvature_direction (in module *SurfaceEvaluation*), 337
max_curvature_direction (in module *TorusEvaluation*), 343
max_edge_length (in module *TessellationOptions*), 263
MAX_MESSAGE_LENGTH (in module *ansys.geometry.core.connection.defaults*), 44
measurement_tools (in module *Modeler*), 448
message (in module *GeometryIssue*), 398
message_id (in module *GeometryIssue*), 397
message_type (in module *GeometryIssue*), 397
MIDDLE (in module *MidSurfaceOffsetType*), 125
min_backend_version() (in module *ansys.geometry.core.misc.checks*), 253
min_bbox (in module *FaceLoop*), 168
min_corner (in module *BoundingBox*), 215
min_curvature (in module *ConeEvaluation*), 309
min_curvature (in module *CylinderEvaluation*), 315
min_curvature (in module *NURBSSurfaceEvaluation*), 323
min_curvature (in module *PlaneEvaluation*), 327
min_curvature (in module *SphereEvaluation*), 333
min_curvature (in module *SurfaceEvaluation*), 336
min_curvature (in module *TorusEvaluation*), 342
min_curvature_direction (in module *ConeEvaluation*), 309
min_curvature_direction (in module *CylinderEvaluation*), 316
min_curvature_direction (in module *NURBSSurfaceEvaluation*), 323

- `min_curvature_direction` (in module *PlaneEvaluation*), 327
- `min_curvature_direction` (in module *SphereEvaluation*), 333
- `min_curvature_direction` (in module *SurfaceEvaluation*), 336
- `min_curvature_direction` (in module *TorusEvaluation*), 342
- `min_distance_between_objects()` (in module *MeasurementTools*), 400
- `minor_radius` (in module *Ellipse*), 286
- `minor_radius` (in module *Torus*), 339
- `mirror()` (in module *Body*), 118
- `mirror()` (in module *IBody*), 90
- `mirror()` (in module *MasterBody*), 104
- `mirrored_copy()` (in module *Circle*), 279
- `mirrored_copy()` (in module *Cone*), 306
- `mirrored_copy()` (in module *Cylinder*), 313
- `mirrored_copy()` (in module *Ellipse*), 287
- `mirrored_copy()` (in module *Sphere*), 330
- `mirrored_copy()` (in module *Torus*), 339
- `modified_bodies` (in module *RepairToolMessage*), 422
- `modify_circular_pattern()` (in module *GeometryCommands*), 184
- `modify_linear_pattern()` (in module *GeometryCommands*), 182
- `modify_placement()` (in module *Component*), 132
- module
 - `ansys.geometry.core`, 35
 - `ansys.geometry.core.connection`, 36
 - `ansys.geometry.core.connection.backend`, 36
 - `ansys.geometry.core.connection.client`, 39
 - `ansys.geometry.core.connection.defaults`, 44
 - `ansys.geometry.core.connection.docker_instance`, 44
 - `ansys.geometry.core.connection.launcher`, 49
 - `ansys.geometry.core.connection.product_instance`, 66
 - `ansys.geometry.core.connection.validate`, 73
 - `ansys.geometry.core.designer`, 74
 - `ansys.geometry.core.designer.beam`, 74
 - `ansys.geometry.core.designer.body`, 82
 - `ansys.geometry.core.designer.component`, 127
 - `ansys.geometry.core.designer.coordinate_system`, 148
 - `ansys.geometry.core.designer.datumplane`, 150
 - `ansys.geometry.core.designer.design`, 152
 - `ansys.geometry.core.designer.designpoint`, 162
 - `ansys.geometry.core.designer.edge`, 164
 - `ansys.geometry.core.designer.face`, 167
 - `ansys.geometry.core.designer.geometry_commands`, 175
 - `ansys.geometry.core.designer.mating_conditions`, 199
 - `ansys.geometry.core.designer.part`, 202
 - `ansys.geometry.core.designer.selection`, 204
 - `ansys.geometry.core.designer.vertex`, 208
 - `ansys.geometry.core.errors`, 435
 - `ansys.geometry.core.logger`, 436
 - `ansys.geometry.core.materials`, 210
 - `ansys.geometry.core.materials.material`, 210
 - `ansys.geometry.core.materials.property`, 211
 - `ansys.geometry.core.math`, 213
 - `ansys.geometry.core.math.bbox`, 214
 - `ansys.geometry.core.math.constants`, 216
 - `ansys.geometry.core.math.frame`, 217
 - `ansys.geometry.core.math.matrix`, 220
 - `ansys.geometry.core.math.misc`, 224
 - `ansys.geometry.core.math.plane`, 226
 - `ansys.geometry.core.math.point`, 228
 - `ansys.geometry.core.math.vector`, 232
 - `ansys.geometry.core.misc`, 240
 - `ansys.geometry.core.misc.accuracy`, 240
 - `ansys.geometry.core.misc.auxiliary`, 244
 - `ansys.geometry.core.misc.checks`, 249
 - `ansys.geometry.core.misc.measurements`, 255
 - `ansys.geometry.core.misc.options`, 260
 - `ansys.geometry.core.misc.units`, 263
 - `ansys.geometry.core.modeler`, 446
 - `ansys.geometry.core.parameters`, 265
 - `ansys.geometry.core.parameters.parameter`, 265
 - `ansys.geometry.core.plotting`, 268
 - `ansys.geometry.core.plotting.plotter`, 270
 - `ansys.geometry.core.plotting.widgets`, 269
 - `ansys.geometry.core.plotting.widgets.show_design_point`, 269
 - `ansys.geometry.core.shapes`, 276
 - `ansys.geometry.core.shapes.box_uv`, 347
 - `ansys.geometry.core.shapes.curves`, 276
 - `ansys.geometry.core.shapes.curves.circle`, 276
 - `ansys.geometry.core.shapes.curves.curve`, 281
 - `ansys.geometry.core.shapes.curves.curve_evaluation`, 283

ansys.geometry.core.shapes.curves.ellipse, 421
 284
 ansys.geometry.core.shapes.curves.line, 423
 290
 ansys.geometry.core.shapes.curves.nurbs, 431
 294
 ansys.geometry.core.shapes.curves.trimmed_nurbs (in module *SpurGear*), 372
 299
 ansys.geometry.core.shapes.parameterization, 372
 349
 ansys.geometry.core.shapes.surfaces, 303
 ansys.geometry.core.shapes.surfaces.cone, 432
 304
 ansys.geometry.core.shapes.surfaces.cylinder, 198
 310
 ansys.geometry.core.shapes.surfaces.nurbs, 198
 316
 ansys.geometry.core.shapes.surfaces.plane, 198
 323
 ansys.geometry.core.shapes.surfaces.sphere, 198
 327
 ansys.geometry.core.shapes.surfaces.surface, 198
 333
 ansys.geometry.core.shapes.surfaces.surface_evaluation, 198
 335
 ansys.geometry.core.shapes.surfaces.torus, 198
 337
 ansys.geometry.core.shapes.surfaces.trimmed_surface, 198
 343
 ansys.geometry.core.sketch, 358
 ansys.geometry.core.sketch.arc, 358
 ansys.geometry.core.sketch.box, 361
 ansys.geometry.core.sketch.circle, 363
 ansys.geometry.core.sketch.edge, 365
 ansys.geometry.core.sketch.ellipse, 366
 ansys.geometry.core.sketch.face, 368
 ansys.geometry.core.sketch.gears, 370
 ansys.geometry.core.sketch.nurbs, 373
 ansys.geometry.core.sketch.polygon, 375
 ansys.geometry.core.sketch.segment, 377
 ansys.geometry.core.sketch.sketch, 379
 ansys.geometry.core.sketch.slot, 392
 ansys.geometry.core.sketch.trapezoid, 393
 ansys.geometry.core.sketch.triangle, 395
 ansys.geometry.core.tools, 397
 ansys.geometry.core.tools.check_geometry, 397
 ansys.geometry.core.tools.measurement_tools, 399
 399
 ansys.geometry.core.tools.prepare_tools, 401
 401
 ansys.geometry.core.tools.problem_areas, 409
 409
 ansys.geometry.core.tools.repair_tool_message, 409
 409
 ansys.geometry.core.tools.repair_tools, 423
 423
 ansys.geometry.core.tools.unsupported, 431
 431
 ansys.geometry.core.typing, 452
 452
 n_sides (in module *Polygon*), 376
 n_teeth (in module *SpurGear*), 372
 name (in module *BeamProfile*), 75
 name (in module *Body*), 112
 name (in module *Component*), 131
 name (in module *CoordinateSystem*), 149
 name (in module *DatumPlane*), 151
 name (in module *DesignPoint*), 163
 name (in module *MasterBody*), 98
 name (in module *MasterComponent*), 204
 name (in module *Material*), 211
 name (in module *MaterialProperty*), 212
 name (in module *NamedSelection*), 206
 name (in module *Parameter*), 266
 name (in module *Part*), 203
 name (in module *SweepWithGuideData*), 128
 name() (in module *IBody*), 84
 named_selections (in module *Design*), 154
 NEGATIVE (in module *ExtrusionDirection*), 148
 NEW_SESSION_HEADER (in module *ansys.geometry.core.logger*), 445
 NO_SPLIT (in module *DraftSide*), 199
 NONE (in module *CollisionType*), 126
 NONE (in module *ExtrudeType*), 197
 norm (in module *Vector2D*), 236
 norm (in module *Vector3D*), 233
 normal (in module *CircleEvaluation*), 281
 normal (in module *ConeEvaluation*), 308
 normal (in module *CylinderEvaluation*), 315
 normal (in module *EllipseEvaluation*), 289
 normal (in module *NURBSSurfaceEvaluation*), 322
 normal (in module *Plane*), 227
 normal (in module *PlaneEvaluation*), 327
 normal (in module *SphereEvaluation*), 332
 normal (in module *SurfaceEvaluation*), 336
 normal (in module *TorusEvaluation*), 341
 normal() (in module *Face*), 171

normal() (in module *ReversedTrimmedSurface*), 346
 normal() (in module *TrimmedSurface*), 344
 normalize() (in module *Vector2D*), 237
 normalize() (in module *Vector3D*), 234
 not_empty (in module *Interval*), 353
 nurbs_from_2d_points() (in module *Sketch*), 386

O

occurrences (in module *MasterComponent*), 204
 offset (in module *AlignCondition*), 200
 OFFSET (in module *FillPatternType*), 198
 offset (in module *OrientCondition*), 201
 offset (in module *TangentCondition*), 201
 offset_edges() (in module *GeometryCommands*), 194
 offset_faces() (in module *GeometryCommands*), 195
 offset_faces_set_radius() (in module *GeometryCommands*), 191
 opacity (in module *Body*), 113
 opacity (in module *Face*), 170
 opacity (in module *MasterBody*), 98
 opacity() (in module *IBody*), 85
 OPAQUE (in module *FillStyle*), 126
 OPEN (in module *ParamForm*), 357
 open_file() (in module *Modeler*), 449
 origin (in module *Circle*), 278
 origin (in module *Cone*), 305
 origin (in module *Cylinder*), 311
 origin (in module *Ellipse*), 286
 origin (in module *Frame*), 219
 origin (in module *Line*), 291
 origin (in module *NURBSSurface*), 318
 origin (in module *PlaneSurface*), 325
 origin (in module *Sphere*), 329
 origin (in module *SpurGear*), 372
 origin (in module *Torus*), 339
 OTHER (in module *DraftSide*), 199
 OTHER (in module *ParamForm*), 357
 OTHER (in module *ParamType*), 357
 OUTER_LOOP (in module *FaceLoopType*), 175
 outer_radius (in module *Polygon*), 376

P

parameter (in module *CircleEvaluation*), 280
 parameter (in module *ConeEvaluation*), 308
 parameter (in module *CurveEvaluation*), 284
 parameter (in module *CylinderEvaluation*), 314
 parameter (in module *EllipseEvaluation*), 289
 parameter (in module *LineEvaluation*), 293
 parameter (in module *NURBSCurveEvaluation*), 298
 parameter (in module *NURBSSurfaceEvaluation*), 322
 parameter (in module *PlaneEvaluation*), 327
 parameter (in module *SphereEvaluation*), 332
 parameter (in module *SurfaceEvaluation*), 336
 parameter (in module *TorusEvaluation*), 341

parameterization() (in module *Circle*), 279
 parameterization() (in module *Cone*), 307
 parameterization() (in module *Curve*), 282
 parameterization() (in module *Cylinder*), 313
 parameterization() (in module *Ellipse*), 288
 parameterization() (in module *Line*), 292
 parameterization() (in module *NURBSCurve*), 297
 parameterization() (in module *NURBSSurface*), 320
 parameterization() (in module *PlaneSurface*), 325
 parameterization() (in module *Sphere*), 330
 parameterization() (in module *Surface*), 334
 parameterization() (in module *Torus*), 340
 parameters (in module *Design*), 154
 PARASOLID_BIN (in module *DesignFileFormat*), 162
 PARASOLID_TEXT (in module *DesignFileFormat*), 162
 parent_component (in module *Beam*), 80
 parent_component (in module *Body*), 113
 parent_component (in module *Component*), 131
 parent_component (in module *CoordinateSystem*), 149
 parent_component (in module *DatumPlane*), 151
 parent_component (in module *DesignPoint*), 163
 parent_id (in module *SweepWithGuideData*), 128
 parse_input() (in module *ApiVersions*), 39
 part (in module *MasterComponent*), 204
 path (in module *SweepWithGuideData*), 128
 perimeter (in module *Box*), 362
 perimeter (in module *Circle*), 278
 perimeter (in module *Ellipse*), 287
 perimeter (in module *Polygon*), 376
 perimeter (in module *SketchCircle*), 364
 perimeter (in module *SketchEllipse*), 368
 perimeter (in module *SketchFace*), 369
 perimeter (in module *Slot*), 393
 PERIODIC (in module *ParamForm*), 357
 PIPE (in module *BeamType*), 81
 plane (in module *PlaneEvaluation*), 327
 plane (in module *Sketch*), 380
 plane_change() (in module *SketchCircle*), 364
 plane_change() (in module *SketchEdge*), 366
 plane_change() (in module *SketchEllipse*), 368
 plane_change() (in module *SketchFace*), 369
 plane_change() (in module *SketchSegment*), 378
 plot() (in module *Body*), 122
 plot() (in module *Component*), 144
 plot() (in module *Face*), 173
 plot() (in module *GeometryPlotter*), 274
 plot() (in module *IBody*), 94
 plot() (in module *MasterBody*), 109
 plot() (in module *Sketch*), 391
 plot_iter() (in module *GeometryPlotter*), 274
 plot_selection() (in module *Sketch*), 391
 plotter_helper (in module *ShowDesignPoints*), 270
 PMDB (in module *DesignFileFormat*), 162
 PNAME (in module *PersistentIdType*), 435

- `point()` (in module *Face*), 171
- `point1` (in module *Triangle*), 396
- `point2` (in module *Triangle*), 396
- `point3` (in module *Triangle*), 396
- `POISSON_RATIO` (in module *MaterialPropertyType*), 213
- `POLYDATA_COLOR_CYCLER` (in module *ansys.geometry.core.plotting.plotter*), 276
- `polygon()` (in module *Sketch*), 389
- `position` (in module *CircleEvaluation*), 280
- `position` (in module *ConeEvaluation*), 308
- `position` (in module *CurveEvaluation*), 284
- `position` (in module *CylinderEvaluation*), 315
- `position` (in module *EllipseEvaluation*), 289
- `position` (in module *LineEvaluation*), 293
- `position` (in module *NURBSCurveEvaluation*), 298
- `position` (in module *NURBSSurfaceEvaluation*), 322
- `position` (in module *PlaneEvaluation*), 327
- `position` (in module *Point3D*), 231
- `position` (in module *SphereEvaluation*), 332
- `position` (in module *SurfaceEvaluation*), 336
- `position` (in module *TorusEvaluation*), 341
- `POSITIVE` (in module *ExtrusionDirection*), 148
- `prepare_and_start_backend()` (in module *ansys.geometry.core.connection.product_instance*), 68
- `prepare_file_for_server_upload()` (in module *ansys.geometry.core.misc.auxiliary*), 249
- `prepare_tools` (in module *Modeler*), 448
- `pressure_angle` (in module *SpurGear*), 372
- `PRIME_ID` (in module *PersistentIdType*), 435
- `process()` (in module *PyGeometryCustomAdapter*), 438
- `profile` (in module *Beam*), 80
- `project_curves()` (in module *Body*), 115
- `project_curves()` (in module *IBody*), 88
- `project_curves()` (in module *MasterBody*), 101
- `project_point()` (in module *Circle*), 279
- `project_point()` (in module *Cone*), 306
- `project_point()` (in module *Curve*), 282
- `project_point()` (in module *Cylinder*), 313
- `project_point()` (in module *Ellipse*), 287
- `project_point()` (in module *Line*), 292
- `project_point()` (in module *NURBSCurve*), 297
- `project_point()` (in module *NURBSSurface*), 321
- `project_point()` (in module *PlaneSurface*), 326
- `project_point()` (in module *ReversedTrimmedSurface*), 346
- `project_point()` (in module *Sphere*), 330
- `project_point()` (in module *Surface*), 334
- `project_point()` (in module *Torus*), 340
- `project_point()` (in module *TrimmedSurface*), 345
- `properties` (in module *Material*), 211
- `proportion()` (in module *BoxUV*), 348
- `protect_grpc()` (in module *ansys.geometry.core.errors*), 436
- Q**
- `quantity` (in module *MaterialProperty*), 212
- R**
- `radius` (in module *Arc*), 359
- `radius` (in module *BeamCircularProfile*), 76
- `radius` (in module *Circle*), 278
- `radius` (in module *Cone*), 305
- `radius` (in module *Cylinder*), 311
- `radius` (in module *Sphere*), 329
- `read_existing_design()` (in module *Modeler*), 449
- `Real` (in module *ansys.geometry.core.typing*), 452
- `RealSequence` (in module *ansys.geometry.core.typing*), 452
- `ref_diameter` (in module *SpurGear*), 372
- `remove_assigned_material()` (in module *Body*), 115
- `remove_assigned_material()` (in module *IBody*), 87
- `remove_assigned_material()` (in module *MasterBody*), 100
- `remove_faces()` (in module *Body*), 122
- `remove_faces()` (in module *IBody*), 93
- `remove_faces()` (in module *MasterBody*), 108
- `remove_material()` (in module *Design*), 154
- `remove_members()` (in module *NamedSelection*), 207
- `remove_rounds()` (in module *PrepareTools*), 403
- `rename_object()` (in module *GeometryCommands*), 181
- `repair()` (in module *InspectResult*), 398
- `repair_geometry()` (in module *RepairTools*), 431
- `repair_tools` (in module *Modeler*), 448
- `repaired` (in module *RepairToolMessage*), 422
- `replace_face()` (in module *GeometryCommands*), 188
- `reset_placement()` (in module *Component*), 132
- `reset_tessellation_cache()` (in module *Body*), 114
- `reset_tessellation_cache()` (in module *MasterBody*), 100
- `revolve_edges()` (in module *GeometryCommands*), 195
- `revolve_faces()` (in module *GeometryCommands*), 186
- `revolve_faces_by_helix()` (in module *GeometryCommands*), 188
- `revolve_faces_up_to()` (in module *GeometryCommands*), 187
- `revolve_sketch()` (in module *Component*), 135
- `RightCenter` (in module *LocationUV*), 349
- `root_diameter` (in module *SpurGear*), 372
- `rotate()` (in module *Body*), 117
- `rotate()` (in module *IBody*), 89
- `rotate()` (in module *MasterBody*), 103
- `rotate()` (in module *TrimmedCurve*), 301
- `rotate_vector()` (in module *Vector3D*), 234
- `run_discovery_script_file()` (in module *Modeler*), 450

`run_if_graphics_required()` (in module `ansys.geometry.core.misc.checks`), 254

S

`save()` (in module `Design`), 154

`scale()` (in module `Body`), 118

`scale()` (in module `IBody`), 89

`scale()` (in module `MasterBody`), 104

`SCDOCX` (in module `DesignFileFormat`), 162

`search_beam()` (in module `Component`), 143

`search_body()` (in module `Component`), 142

`search_component()` (in module `Component`), 142

`search_plane()` (in module `Component`), 143

`second_derivative` (in module `CircleEvaluation`), 281

`second_derivative` (in module `CurveEvaluation`), 284

`second_derivative` (in module `EllipseEvaluation`), 290

`second_derivative` (in module `LineEvaluation`), 294

`second_derivative` (in module `NURBSCurveEvaluation`), 299

`section_anchor` (in module `BeamCrossSectionInfo`), 77

`section_angle` (in module `BeamCrossSectionInfo`), 77

`section_frame` (in module `BeamCrossSectionInfo`), 77

`section_profile` (in module `BeamCrossSectionInfo`), 77

`sector_area` (in module `Arc`), 360

`segment()` (in module `Sketch`), 382

`segment_from_point_and_vector()` (in module `Sketch`), 383

`segment_from_vector()` (in module `Sketch`), 383

`segment_to_point()` (in module `Sketch`), 382

`select()` (in module `Sketch`), 382

`self_intersect()` (in module `Interval`), 354

`self_unite()` (in module `Interval`), 354

`semi_latus_rectum` (in module `Ellipse`), 287

`SERVER_ANGLE` (in module `DefaultUnitsClass`), 257

`SERVER_AREA` (in module `DefaultUnitsClass`), 257

`SERVER_LENGTH` (in module `DefaultUnitsClass`), 257

`SERVER_VOLUME` (in module `DefaultUnitsClass`), 257

`services` (in module `GrpcClient`), 42

`set_color()` (in module `Body`), 117

`set_color()` (in module `Face`), 171

`set_color()` (in module `IBody`), 85

`set_color()` (in module `MasterBody`), 103

`set_export_id()` (in module `UnsupportedCommands`), 433

`set_fill_style()` (in module `Body`), 117

`set_fill_style()` (in module `IBody`), 84

`set_fill_style()` (in module `MasterBody`), 103

`set_multiple_export_ids()` (in module `UnsupportedCommands`), 433

`set_name()` (in module `Body`), 117

`set_name()` (in module `Component`), 132

`set_name()` (in module `IBody`), 84

`set_name()` (in module `MasterBody`), 103

`set_opacity()` (in module `Face`), 171

`set_opacity()` (in module `MasterBody`), 103

`set_parameter()` (in module `Design`), 159

`set_shared_topology()` (in module `Component`), 133

`set_shared_topology()` (in module `Design`), 158

`set_suppressed()` (in module `Body`), 117

`set_suppressed()` (in module `IBody`), 85

`set_suppressed()` (in module `MasterBody`), 103

`setLevel()` (in module `Logger`), 442

`setLevel()` (in module `PyGeometryCustomAdapter`), 438

`setup_offset_relationship()` (in module `Face`), 172

`shape` (in module `Edge`), 165

`shape` (in module `Face`), 170

`share_topology()` (in module `PrepareTools`), 404

`shared_topology` (in module `Component`), 131

`SHARETYPE_GROUPS` (in module `SharedTopologyType`), 147

`SHARETYPE_MERGE` (in module `SharedTopologyType`), 147

`SHARETYPE_NONE` (in module `SharedTopologyType`), 147

`SHARETYPE_SHARE` (in module `SharedTopologyType`), 147

`shear_center` (in module `BeamProperties`), 79

`SHEAR_CENTER` (in module `SectionAnchorType`), 82

`SHEAR_MODULUS` (in module `MaterialPropertyType`), 213

`shell_body()` (in module `Body`), 122

`shell_body()` (in module `IBody`), 93

`shell_body()` (in module `MasterBody`), 108

`show()` (in module `GeometryPlotter`), 275

`SIGINT_TRACKER` (in module `ansys.geometry.core.errors`), 436

`sketch` (in module `SweepWithGuideData`), 128

`sketch_polydata()` (in module `Sketch`), 391

`sketch_polydata_edges()` (in module `Sketch`), 391

`sketch_polydata_faces()` (in module `Sketch`), 391

`SketchObject` (in module `ansys.geometry.core.sketch.sketch`), 392

`SKEWED` (in module `FillPatternType`), 198

`slot()` (in module `Sketch`), 388

`SPACECLAIM` (in module `BackendType`), 37

`SPACECLAIM_EXE` (in module `ansys.geometry.core.connection.product_instance`), 72

`SPACECLAIM_FOLDER` (in module `ansys.geometry.core.connection.product_instance`), 71

`SPECIFIC_HEAT` (in module `MaterialPropertyType`), 213

`sphere` (in module `SphereEvaluation`), 332

`split_body()` (in module `GeometryCommands`), 189

`SPRING` (in module `BeamType`), 81

`spur_gear()` (in module `Sketch`), 390

`start` (in module `Arc`), 359

`start` (in module `Beam`), 80

- start (in module *Edge*), 166
- start (in module *Interval*), 353
- start (in module *SketchEdge*), 365
- start (in module *SketchNurbs*), 374
- start (in module *SketchSegment*), 378
- start (in module *TrimmedCurve*), 300
- std_out_handler (in module *Logger*), 441
- std_out_handler (in module *PyGeometryCustom-Adapter*), 438
- stdout_handler (in module *PyGeometryCustom-Adapter*), 438
- STDOUT_MSG_FORMAT (in module *ansys.geometry.core.logger*), 445
- STEP (in module *DesignFileFormat*), 162
- STRIDE (in module *DesignFileFormat*), 162
- string_to_loglevel (in module *ansys.geometry.core.logger*), 446
- subtract() (in module *Body*), 124
- subtract() (in module *IBody*), 95
- subtract() (in module *MasterBody*), 110
- subtract_bodies (in module *EnclosureOptions*), 401
- SUCCESS (in module *ParameterUpdateStatus*), 268
- success (in module *RepairToolMessage*), 422
- surface (in module *NURBSSurfaceEvaluation*), 322
- surface_area (in module *Cone*), 306
- surface_area (in module *Sphere*), 329
- surface_area (in module *Torus*), 339
- surface_area() (in module *Cylinder*), 312
- surface_deviation (in module *TessellationOptions*), 263
- surface_offset (in module *Body*), 113
- surface_offset (in module *MasterBody*), 99
- surface_offset() (in module *IBody*), 86
- surface_thickness (in module *Body*), 113
- surface_thickness (in module *MasterBody*), 99
- surface_thickness() (in module *IBody*), 86
- surface_type (in module *Face*), 170
- SURFACETYPE_CONE (in module *SurfaceType*), 174
- SURFACETYPE_CYLINDER (in module *SurfaceType*), 174
- SURFACETYPE_NURBS (in module *SurfaceType*), 174
- SURFACETYPE_PLANE (in module *SurfaceType*), 174
- SURFACETYPE_PROCEDURAL (in module *SurfaceType*), 174
- SURFACETYPE_SPHERE (in module *SurfaceType*), 174
- SURFACETYPE_TORUS (in module *SurfaceType*), 174
- SURFACETYPE_UNKNOWN (in module *SurfaceType*), 174
- sweep_chain() (in module *Component*), 134
- sweep_sketch() (in module *Component*), 134
- sweep_with_guide() (in module *Component*), 135
- tangent (in module *CircleEvaluation*), 281
- tangent (in module *EllipseEvaluation*), 289
- tangent (in module *LineEvaluation*), 293
- target() (in module *GrpcClient*), 42
- TENSILE_STRENGTH (in module *MaterialPropertyType*), 213
- tessellate() (in module *Body*), 120
- tessellate() (in module *Component*), 144
- tessellate() (in module *Face*), 172
- tessellate() (in module *IBody*), 91
- tessellate() (in module *MasterBody*), 106
- THERMAL_CONDUCTIVITY (in module *MaterialPropertyType*), 213
- THERMALFLUID (in module *BeamType*), 81
- thicken_faces() (in module *GeometryCommands*), 194
- THIS (in module *DraftSide*), 199
- tight_tolerance (in module *SweepWithGuideData*), 128
- tip_diameter (in module *SpurGear*), 372
- to_dict() (in module *ImportOptions*), 261
- to_dict() (in module *ImportOptionsDefinitions*), 262
- TOP (in module *MidSurfaceOffsetType*), 125
- TopCenter (in module *LocationUV*), 349
- TopLeft (in module *LocationUV*), 349
- TopRight (in module *LocationUV*), 349
- torsion_constant (in module *BeamProperties*), 79
- torus (in module *TorusEvaluation*), 341
- TOUCH (in module *CollisionType*), 126
- transform (in module *MasterComponent*), 204
- transform() (in module *Point3D*), 231
- transform() (in module *Vector3D*), 234
- transform_point2d_local_to_global() (in module *Frame*), 219
- transformation_matrix (in module *Frame*), 219
- transformed_copy() (in module *Circle*), 279
- transformed_copy() (in module *Cone*), 306
- transformed_copy() (in module *Curve*), 282
- transformed_copy() (in module *Cylinder*), 312
- transformed_copy() (in module *Ellipse*), 288
- transformed_copy() (in module *Line*), 292
- transformed_copy() (in module *NURBSCurve*), 297
- transformed_copy() (in module *NURBSSurface*), 320
- transformed_copy() (in module *PlaneSurface*), 326
- transformed_copy() (in module *Sphere*), 330
- transformed_copy() (in module *Surface*), 334
- transformed_copy() (in module *Torus*), 339
- transformed_copy() (in module *TrimmedCurve*), 301
- translate() (in module *Body*), 117
- translate() (in module *IBody*), 89
- translate() (in module *MasterBody*), 102
- translate() (in module *TrimmedCurve*), 301
- translate_bodies() (in module *Component*), 140
- translate_sketch_plane() (in module *Sketch*), 380
- translate_sketch_plane_by_distance() (in module *Sketch*), 381

- translate_sketch_plane_by_offset() (in module Sketch), 381
- TRANSPARENT (in module FillStyle), 126
- transport_mode (in module LocalDockerInstance), 46
- trapezoid() (in module Sketch), 387
- tree_print() (in module Component), 145
- triangle() (in module Sketch), 386
- trim() (in module Curve), 283
- trim() (in module Surface), 335
- type (in module FaceLoop), 168
- type (in module MaterialProperty), 212
- type (in module Parameterization), 356
- ## U
- u (in module ParamUV), 350
- u_derivative (in module ConeEvaluation), 308
- u_derivative (in module CylinderEvaluation), 315
- u_derivative (in module NURBSSurfaceEvaluation), 322
- u_derivative (in module PlaneEvaluation), 327
- u_derivative (in module SphereEvaluation), 332
- u_derivative (in module SurfaceEvaluation), 336
- u_derivative (in module TorusEvaluation), 342
- unit (in module PhysicalQuantity), 264
- unit() (in module Point2D), 229
- unit() (in module Point3D), 231
- UNIT_MAP (in module ansys.geometry.core.parameters.parameter), 268
- unite() (in module Body), 124
- unite() (in module IBody), 95
- unite() (in module Interval), 354
- unite() (in module MasterBody), 110
- UNITS (in module ansys.geometry.core.misc.units), 264
- UNITVECTOR2D_X (in module ansys.geometry.core.math.constants), 217
- UNITVECTOR2D_Y (in module ansys.geometry.core.math.constants), 217
- UNITVECTOR3D_X (in module ansys.geometry.core.math.constants), 217
- UNITVECTOR3D_Y (in module ansys.geometry.core.math.constants), 217
- UNITVECTOR3D_Z (in module ansys.geometry.core.math.constants), 217
- UNKNOWN (in module BeamType), 81
- UNKNOWN (in module ParameterUpdateStatus), 268
- unsupported (in module Modeler), 448
- update() (in module ShowDesignPoints), 270
- update_fill_pattern() (in module GeometryCommands), 186
- USE_SERVICE_COLORS (in module ansys.geometry.core), 453
- use_service_colors (in module GeometryPlotter), 271
- USE_TRACKER_TO_UPDATE_DESIGN (in module ansys.geometry.core), 453
- uu_derivative (in module ConeEvaluation), 309
- uu_derivative (in module CylinderEvaluation), 315
- uu_derivative (in module NURBSSurfaceEvaluation), 322
- uu_derivative (in module PlaneEvaluation), 327
- uu_derivative (in module SphereEvaluation), 332
- uu_derivative (in module SurfaceEvaluation), 336
- uu_derivative (in module TorusEvaluation), 342
- uv_derivative (in module ConeEvaluation), 309
- uv_derivative (in module CylinderEvaluation), 315
- uv_derivative (in module NURBSSurfaceEvaluation), 322
- uv_derivative (in module PlaneEvaluation), 327
- uv_derivative (in module SphereEvaluation), 332
- uv_derivative (in module SurfaceEvaluation), 336
- uv_derivative (in module TorusEvaluation), 342
- ## V
- v (in module ParamUV), 350
- V_21 (in module ApiVersions), 38
- V_22 (in module ApiVersions), 38
- V_231 (in module ApiVersions), 38
- V_232 (in module ApiVersions), 38
- V_241 (in module ApiVersions), 38
- V_242 (in module ApiVersions), 38
- V_251 (in module ApiVersions), 38
- V_252 (in module ApiVersions), 38
- V_261 (in module ApiVersions), 39
- v_derivative (in module ConeEvaluation), 308
- v_derivative (in module CylinderEvaluation), 315
- v_derivative (in module NURBSSurfaceEvaluation), 322
- v_derivative (in module PlaneEvaluation), 327
- v_derivative (in module SphereEvaluation), 332
- v_derivative (in module SurfaceEvaluation), 336
- v_derivative (in module TorusEvaluation), 342
- validate() (in module ansys.geometry.core.connection.validate), 73
- value (in module DatumPlane), 151
- value (in module DesignPoint), 163
- value (in module ExportIdData), 432
- value (in module Measurement), 258
- VARIABLE (in module MidSurfaceOffsetType), 125
- vertices (in module Body), 113
- vertices (in module Edge), 166
- vertices (in module Face), 170
- vertices (in module MasterBody), 99
- vertices (in module NamedSelection), 207
- vertices() (in module IBody), 85
- visualization_polydata (in module Arc), 360
- visualization_polydata (in module Box), 363
- visualization_polydata (in module Cone), 306

[visualization_polydata \(in module Cylinder\)](#), 312
[visualization_polydata \(in module Gear\)](#), 370
[visualization_polydata \(in module NURBSSurface\)](#), 318
[visualization_polydata \(in module PlaneSurface\)](#), 325
[visualization_polydata \(in module Polygon\)](#), 376
[visualization_polydata \(in module SketchCircle\)](#), 364
[visualization_polydata \(in module SketchEdge\)](#), 366
[visualization_polydata \(in module SketchEllipse\)](#), 368
[visualization_polydata \(in module SketchFace\)](#), 369
[visualization_polydata \(in module SketchNurbs\)](#), 374
[visualization_polydata \(in module SketchSegment\)](#), 378
[visualization_polydata \(in module Slot\)](#), 393
[visualization_polydata \(in module Sphere\)](#), 329
[visualization_polydata \(in module Torus\)](#), 339
[visualization_polydata \(in module Trapezoid\)](#), 395
[visualization_polydata \(in module Triangle\)](#), 396
[visualization_polydata\(\)](#) (in module Surface), 334
[volume \(in module Body\)](#), 113
[volume \(in module Cone\)](#), 306
[VOLUME \(in module DefaultUnitsClass\)](#), 257
[volume \(in module MasterBody\)](#), 99
[volume \(in module Sphere\)](#), 329
[volume \(in module Torus\)](#), 339
[volume\(\)](#) (in module Cylinder), 312
[volume\(\)](#) (in module IBody), 86
[vv_derivative \(in module ConeEvaluation\)](#), 309
[vv_derivative \(in module CylinderEvaluation\)](#), 315
[vv_derivative \(in module NURBSSurfaceEvaluation\)](#), 322
[vv_derivative \(in module PlaneEvaluation\)](#), 327
[vv_derivative \(in module SphereEvaluation\)](#), 332
[vv_derivative \(in module SurfaceEvaluation\)](#), 336
[vv_derivative \(in module TorusEvaluation\)](#), 342

W

[wait_until_healthy\(\)](#) (in module *ansys.geometry.core.connection.client*), 43
[WARN \(in module ansys.geometry.core.logger\)](#), 445
[warning \(in module Logger\)](#), 441
[warping_constant \(in module BeamProperties\)](#), 79
[watertight \(in module TessellationOptions\)](#), 263
[weights \(in module NURBSCurve\)](#), 296
[weights \(in module NURBSSurface\)](#), 318
[weights \(in module SketchNurbs\)](#), 374
[width \(in module Box\)](#), 362
[width \(in module Slot\)](#), 393
[WINDOWS_24_1 \(in module GeometryContainers\)](#), 48
[WINDOWS_24_2 \(in module GeometryContainers\)](#), 48

[WINDOWS_25_1 \(in module GeometryContainers\)](#), 48
[WINDOWS_25_2 \(in module GeometryContainers\)](#), 48
[WINDOWS_GEOMETRY_SERVICE_FOLDER \(in module ansys.geometry.core.connection.product_instance\)](#), 71
[WINDOWS_LATEST \(in module GeometryContainers\)](#), 48
[WINDOWS_LATEST_UNSTABLE \(in module GeometryContainers\)](#), 48
[WINDOWS_SERVICE \(in module BackendType\)](#), 37

X

[x \(in module Point2D\)](#), 229
[x \(in module Point3D\)](#), 231
[x \(in module Vector2D\)](#), 236
[x \(in module Vector3D\)](#), 233

Y

[y \(in module Point2D\)](#), 229
[y \(in module Point3D\)](#), 231
[y \(in module Vector2D\)](#), 236
[y \(in module Vector3D\)](#), 233

Z

[z \(in module Point3D\)](#), 231
[z \(in module Vector3D\)](#), 233
[ZERO_POINT2D \(in module *ansys.geometry.core.math.constants*\)](#), 217
[ZERO_POINT3D \(in module *ansys.geometry.core.math.constants*\)](#), 217
[ZERO_VECTOR2D \(in module *ansys.geometry.core.math.constants*\)](#), 217
[ZERO_VECTOR3D \(in module *ansys.geometry.core.math.constants*\)](#), 217