CAD Integration

Ansys

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

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CAD Integration Overview

Ansys Workbench is a CAD-neutral environment that supports bidirectional, direct, and associative geometry interfaces (plug-ins (p. 133)) with CAD systems as well as non-associative interfaces (readers) that generally do not require the CAD be installed. The readers (p. 133) and associative geometry interfaces (p. 133) are *not* interchangeable. The specific import options that are supported can be viewed at Geometry Preferences (p. 8).

The plug-ins support import/update without translation to the intermediate geometry formats. The associative geometry interfaces allow you to make parametric changes in a CAD system or drive those changes from within Ansys Workbench and when the geometry is updated assigned scopings will persist if the topology is present in the updated model. The Named Selection Manager (p. 25), available in most integrated CAD systems, provides a means to create custom selections within the CAD systems for use in modeling, meshing, and analysis.

Readers, with the exception of the CAPRI interface, do not require the CAD system to be present to import geometry files. These interfaces however, are not associative, nor bi-directionally parametric.

Ansys CAD integration supports the Smart CAD Update, where supported by the CAD, and Selective Update of CAD parts instead of updating an entire model. All interfaces can update the model using Compare Parts on Update (p. 22) and those parts that are not modified will maintain their existing mesh so it does not need to be regenerated.

Bidirectional CAD Connections

AutoCAD (*.dwg, *.dxf) (p. 34)
CATIA V5 Associative Geometry Interface (*.CATPart, *.CATProduct) (p. 53)
Creo Elements/Direct Modeling (*.pkg, *.bdl, *.ses, *.sda, *.sdp, *.sdac, *.sdpc) (p. 59)
Creo Parametric Associative Geometry Interface (*.prt, *.asm) (p. 66)
Autodesk Inventor Associative Geometry Interface (*.ipt, *.iam) (p. 43)
Fusion (*.f3d, *.f3z) (p. 80)
NX Associative Geometry Interface (*.prt) (p. 90)
Solid Edge (*.par, *.asm, *.psm, *.pwd) (p. 101)
SOLIDWORKS Associative Geometry Interface (*.sldprt, *.sldasm) (p. 111)

CAD Readers

ACIS (*.sat, *.sab) (p. 32)
AutoCAD Reader (*.dwg, *.dxf) (p. 35)
Autodesk Inventor Reader (*.ipt, *.iam) (p. 40)
CATIA V4 Reader (*.model, *.exp, *.session) (p. 47)

CATIA V5 Reader (*.CATPart, *.CATProduct) (p. 49)
CATIA V6 Reader (*.3dxml) (p. 57)
Creo Parametric Reader (*.prt, *.prt.*, *.asm, *.asm.*) (p. 62)
IGES (*.igs, *.iges) (p. 82)
JT Reader (*.jt) (p. 84)
Monte Carlo N-Particle (*.mcnp) (p. 86)
NX Reader (*.prt) (p. 87)
Parasolid (*.x_t, *.xmt_txt, *.x_b, *.xmt_bin) (p. 95)
Revit (*.rvt, *.rfa) (p. 98)
Rhinoceros (*.3dm) (p. 99)
Solid Edge Reader (*.par, *.asm, *.psm) (p. 101)
SOLIDWORKS Reader (*.sldprt, *.sldasm) (p. 108)
STEP (*.stp, *.step) (p. 116)

Geometry Export

IGES (*.igs, *.iges) (p. 82)
Ansys MAPDL (*.anf)
Monte Carlo N-Particle (*.mcnp) (p. 86)
Parasolid (*.x_t, *.xmt_txt, *.x_b, *.xmt_bin) (p. 95)
STEP (*.stp, *.step) (p. 116)

For more detailed information about Ansys CAD:

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Introduction

With the understanding that all engineering simulation is based on geometry to represent the design, there are several methods of accessing CAD models within Ansys Workbench, depending upon the level of integration and the interface products licensed at your site.

Starting from within Ansys Workbench

To start from within Ansys Workbench, double-click **Geometry** in the **Component Systems** toolbox. You can also select **Geometry** in the **Component Systems** toolbox and drag it to the **Project Schematic**. Additionally, any system that contains a geometry cell can be used (for example, Static Structural).

Access via Ansys DesignModeler or Ansys SpaceClaim Direct Modeler

For **Geometry** access via Workbench-integrated applications Ansys DesignModeler or Ansys SpaceClaim Direct Modeler, right-click to select **New Design Modeler Geometry** or **New SpaceClaim Direct Modeler Geometry**, or browse to a geometry file.

Access via CAD system

For **Geometry** access via CAD system with Associative reader, or CAD system with plug-in (p. 133), right-click to select **Import Geometry**, then **Browse** to a geometry file OR select a model from the active documents.

For **Geometry** access via CAD file with reader (p. 133), right-click to select **Import Geometry**, then **Browse** to a geometry file.

Starting from within the CAD System

For **Geometry** access via a CAD system with plug-in (p. 133), start Ansys Workbench by selecting the Ansys pull-down menu from the CAD system's toolbar.

Ansys Workbench will open and automatically create a new **Geometry** system on the **Project Schematic**.

Geometry Interface Support

When installing Ansys Workbench, you may select specific CAD systems in the Ansys Geometry Interfaces section.

For information about past, present and future platform support, see the Platform Support section of the Ansys Website.

Matrices for supported CAD systems on Ansys Workbench at the time of release:

Linux

Windows

Linux

Reader/Plug-In	Versions of CAD Package	Operating System			
		Red Hat 8 and 9	SuSE Linux Enterprise 15	Rocky Linux 8.10 and 9.4	Ubuntu 20.04 and 22.04
		Supported Platform			
		Linux 64 (Intel EM64T, AMD64)			
READERS					
ACIS	2023	√	√	√	√
Autodesk AutoCAD	2019				
CATIA V4	4.2.5				
CATIA V5	V5–6R2024	√	√	√	√
CATIA V5 — (CADNexus CAPRI CAE Gateway V4.10.0)	V5–6R2022, V5–6R023, V5–6R2024				
CATIA V6	R2024x				
Creo Parametric	11.0				
IGES	5.3	√	√	√	√
Autodesk Inventor	2025				
JT	11.5				
Monte Carlo N-Particle		√	√	√	√
NX	2406				
Parasolid	35.1	√	√	√	√
Rhinoceros	8				
Solid Edge	2024				
SOLIDWORKS	2024				
STEP	AP242	√	√	√	√
PLUG-INS/PSEUDO-READER					
Autodesk AutoCAD	2025				
	2024				

Reader/Plug-In	Versions of CAD Package	Operating System			
		Red Hat 8 and 9	SuSE Linux Enterprise 15	Rocky Linux 8.10 and 9.4	Ubuntu 20.04 and 22.04
		Supported Platform			
		Linux 64 (Intel EM64T, AMD64)			
Creo Elements/Direct Modeling	20.7				
	20.6				
Creo Parametric	11.0				
	10.0				
	9.0				
	7.0				
Autodesk Inventor	2025				
	2024				
NX	2406	√3		√2	
	2312	√3	√1		
	2306		√1		
Solid Edge	2024				
	2023				
SOLIDWORKS	2024				
	2023				
	2022				
	2021				

√ = supported

1. support limited to SuSE Linux Enterprise 15
2. support limited to Rocky 8.10
3. support limited to Red Hat 8

Windows

Reader/Plug-In	Versions of CAD Package	Operating System	
		Windows 10	Windows 11
		Supported Platform	
		Intel EM64T, AMD64	
READERS			
ACIS	2023	√	√
Autodesk AutoCAD	2019	√	√
CATIA V4	4.2.5	√	√
CATIA V5	V5–6R2024	√	√
CATIA V5 — (CADNexus CAPRI CAE Gateway V4.10.0)	V5–6R2022, V5–6R2023, V5–6R2024	√	√
CATIA V6	R2024x	√	√
Creo Parametric	11.0	√	√
IGES	5.3	√	√
Autodesk Inventor	2025	√	√
JT	11.5	√	√
Monte Carlo N-Particle [1]		√	
NX	2406	√	√
Parasolid	36.1	√	√
Rhinoceros	8	√	√
Solid Edge	2024	√	√
SOLIDWORKS	2024	√	√
STEP	AP242	√	√
PLUG-INS/PSEUDO-READER			
Autodesk AutoCAD	2025	√	√
	2024	√	√
Creo Elements/Direct Modeling	20.7	√	√
	20.6	√	√
Creo Parametric	11.0	√	√**5**

Reader/Plug-In	Versions of CAD Package	Operating System	
		Windows 10	Windows 11
		Supported Platform	
		Intel EM64T, AMD64	
READERS			
	10.0	√	√5
	9.0	√	√5
	7.0	√6	√4
Fusion	2.0.15995 onward	√	√
Autodesk Inventor	2025	√	√
	2024	√	√
NX	2406	√	√
	2312	√	√
	2306	√	√
Revit	2024	√	√
Solid Edge	2024	√	√
	2023	√	√3
SOLIDWORKS	2024	√2	√2
	2023	√2	√2
	2022	√2	
	2021	√2	
	2020		
SpaceClaim	2020	√	

√ = supported

1. applicable only to Ansys DesignModeler
2. validated by external third party
3. requires 2022 Maintenance Pack
4. version 7.0.8.0; requires Windows 11 Version 21H2 or higher
5. requires Windows 11 Version 21H2 or higher
6. minimum supported version 7.0.4.0

Project Schematic Presence Related to CAD Integration

The **Project Schematic** captures the project and workflow of your project, providing a visual representation of the objects in the project and their relationship to each other.

See CAD in Project Schematic for information about:

- Launching Ansys Workbench from CAD Systems
- Material Processing
- Product & CAD Configuration Manager (p. 24)

Project Schematic Presence topics:

Geometry Preferences

Geometry Preferences

The following geometry preferences are accessible at the locations listed below and may vary in name depending on the location (for example, for **CAD Associativity** in the Workbench Options, the counterpart is **Use Associativity** in the Workbench Project Schematic).

- **Workbench Options**: On the Workbench main menu, navigate to **Tools**> **Options**> **Geometry Import** to set the default values of the preferences. A checked check box indicates enabled or yes, and an unchecked check box indicates disabled or no.
- **Workbench Project Schematic**: Right-click the Geometry cell on a system and choose Properties.

Note:

In addition to their appearances in the Workbench Options and Workbench Project Schematic, read-only indications of the settings are shown in the Details View of the Geometry object in Workbench applications.

The preferences are displayed in two categories:

- **Basic Geometry Options** (p. 9)
- **Advanced Geometry Options** (p. 14)

DesignModeler and SpaceClaim Functionality

When DesignModeler or SpaceClaim is configured in add-in mode (and the geometry model source is DesignModeler or SpaceClaim), most of the import preferences listed below are hidden. Except for the analysis-specific preferences (**Analysis Type** and **Compare Parts On Update**), all the applicable model data in the applications is transferred transparently.

Basic Geometry Options

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
Solid Bodies (check box)	**Solid Bodies** (check box)	Imports solid bodies.[1 (p. 14)] The default is **Yes**. The preference applies to the following: All CAD Associative Geometry Interfaces and Readers.
Surface Bodies (check box)	**Surface Bodies** (check box)	Imports surface bodies.[1 (p. 14)] The default is **Yes**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 34) Associative Geometry Interface and Reader Autodesk Inventor (p. 40) Associative Geometry Interface and Reader (Merged/sewn work surfaces are not supported.) CATIA (p. 46) V4 Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader CATIA V6 (p. 57) Reader Creo Elements/Direct Modeling (p. 59) Creo Parametric (p. 62) Associative Geometry Interface and Reader Fusion (p. 77) Associative Geometry Interface and Reader IGES (p. 82) Reader JT (p. 84) Reader NX (p. 87) Associative Geometry Interface and Reader Parasolid (p. 95) Reader Revit (*.rvt, *.rfa) (p. 98) Solid Edge (p. 101) Associative Geometry Interface and Reader SOLIDWORKS (p. 108) Associative Geometry Interface and Reader STEP (p. 116) Reader

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
Line Bodies (check box)	**Line Bodies** (check box)	Imports line bodies. The default is **No**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 34) Associative Geometry Interface and Reader CATIA (p. 46) V4 Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader CATIA V6 (p. 57) Reader Creo Parametric (p. 62) Associative Geometry Interface and Reader IGES (p. 82) Reader JT (p. 84) Reader NX (p. 87) Associative Geometry Interface and Reader Parasolid (p. 95) Reader Solid Edge (p. 101) Associative Geometry Interface and Reader SOLIDWORKS (p. 108) Associative Geometry Interface STEP (p. 116) Reader Note that only DesignModeler and Ansys SpaceClaim Direct Modeler support cross-section definition/import.
Parameters (drop-down menu)	**Parameters** (drop-down menu)	Controls parameter processing, which can slow down overall import performance. Choices are: **None**: No parameters from the CAD source are processed. **Independent**: Parameters from the CAD source which are independent are processed. These are input parameters which can be modified and promoted as design parameters. **All**: Both independent and dependent parameters are processed from the CAD source. Independent parameters can be modified and promoted as design parameters. Dependent parameters are read-only and cannot be modified or promoted. The default is **Independent**. The preference applies to the following:

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		Autodesk Inventor (p. 40) Associative Geometry Interface CATIA (p. 46) V5 Associative Geometry Interface Creo Parametric (p. 62) Associative Geometry Interface Fusion (p. 77) Associative Geometry Interface and Reader NX (p. 87) Associative Geometry Interface Solid Edge (p. 101) Associative Geometry Interface SOLIDWORKS (p. 108) Associative Geometry Interface
Parameters> Filtering Prefixes and Suffixes (text field)	**Parameter Key** (text field)	When **Parameters** is set to **All** or **Independent**, enables you to specify a key that must appear at the beginning or end of a CAD parameter name for that parameter to be enabled in Workbench. The default is ANS;DS. You may specify multiple prefix/suffix values, with each one separated by a semicolon. For example, if you want to import some parameters that are keyed with DS and others that are keyed with PARAM, enter DS;PARAM. The text field is case-insensitve, so entering ds;param works just as effectively. There is no limit on the number of prefixes/suffixes that can be specified in a single key preference.
CAD Attributes (check box)	**Attributes** (check box)	Allows import of CAD system attributes into the Ansys Mechanical application models. Enable this option to import Motion Loads. The default is **No**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 34) Reader Autodesk Inventor (p. 40) Associative Geometry Interface and Reader CATIA (p. 46) V4 Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader CATIA V6 (p. 57) Reader Creo Parametric (p. 62) Associative Geometry Interface and Reader

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		Fusion (p. 77) Associative Geometry Interface and Reader JT (p. 84) Reader NX (p. 87) Associative Geometry Interface and Reader Revit (*.rvt, *.rfa) (p. 98) Solid Edge (p. 101) Associative Geometry Interface and Reader SOLIDWORKS (p. 108) Associative Geometry Interface and Reader
CAD Attributes> Filtering Prefixes (text field)	**Attribute Key** (text field)	This field can contain any number of prefixes, with each prefix delimited by a semicolon. By default, the filter is set to **SDFEA;DDM**. If the filter is set to an empty string, all applicable entities are imported as CAD system attributes.
Named Selections (check box)	**Named Selections** (check box)	Creates a named selection based on data generated in the CAD system or in the DesignModeler application. You must set the value in the **Filtering Prefixes** field (described below) to the desired value. Upon attaching or updating, a **Named Selection** branch is added to the tree and its name appears in the drop-down display within the **Named Selection Toolbar**. It is maintained as a CAD named selection unless the branch is altered (for example, if entities are added or deleted, or a selection is renamed). After updating, CAD named selections are deleted and replaced with named selections that are imported for the updated model.[2 (p. 14)] The default is **No**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 35) Reader Autodesk Inventor (p. 40) Associative Geometry Interface and Reader CATIA (p. 46) V4 Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader CATIA V6 (p. 57) Reader Creo Elements/Direct Modeling (p. 59) Creo Parametric (p. 62) Associative Geometry Interface and Reader

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		Fusion (p. 77) Associative Geometry Interface and Reader JT (p. 84) Reader NX (p. 87) Associative Geometry Interface and Reader Revit (*.rvt, *.rfa) (p. 98) Solid Edge (p. 101) Associative Geometry Interface and Reader SOLIDWORKS (p. 108) Associative Geometry Interface and Reader
Named Selections> Filtering Prefixes (text field)	**Named Selection Key** (text field)	(Displayed only when Named Selection Processing is set to **Yes** in the Details View.) Allows you to set the named selection processing prefix key. The default is NS. This field can have any number of prefixes with each prefix delimited by a semicolon (for example: `NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded`). By default the filter is set to NS. If the filter is set to an empty string all applicable entities will be imported as named selections.
Material Properties (check box)	**Material Properties** (check box)	Allows import of material data defined in the CAD system. Only a subset of material data will be imported. This will include Young's Modulus, Poisson Ratio, Mass Density, Specific Heat, Thermal Conductivity and Thermal Expansion Coefficient. Limited additional data may be imported depending on CAD support. A material file will be created that reflects each of the CAD materials assigned to the model. You can validate the imported data as well as edit all of the material property values in Engineering Data. Choosing **Update** will allow you to import new materials but will not update values of previously imported materials. This is done to avoid overwriting user changes to previously imported material files. The default is **No**. The preference applies to the following: ACIS (p. 32) Reader Autodesk Inventor (p. 40) Associative Geometry Interface and Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader Creo Parametric (p. 62) Associative Geometry Interface and Reader

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		Fusion (p. 77) Associative Geometry Interface and Reader NX (p. 87) Associative Geometry Interface and Reader Solid Edge (p. 101) Associative Geometry Interface and Reader SOLIDWORKS (p. 108) Associative Geometry Interface and Reader

[1] These preferences are on a per part basis. Parts with solid bodies and surface bodies will result in an attach failure if both import type preferences are selected. For assemblies however, where different components are solely solid body or surface body, import of each part will be successful.

[2] Limitations on importing named selections:

- If you use a CAD system filter for entities, you must be able to create entities with names that correspond to the filter.
- Named selection sets should contain entities of only a single dimension (for example, faces or edges).
- Refer to the Named Selection Import based on Entities (p. 26) table to determine the CAD system support for the various entities (vertex, edge, face, and body).

Advanced Geometry Options

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
Analysis Type (drop-down menu)	**Analysis Type** (drop-down menu)	• In **Options** dialog box, sets the default for the **Project Schematic** setting. • On **Project Schematic**, sets the geometry for a 2-D simulation or a 3-D simulation. • In Details View, provides a read-only indication of the current analysis type. The preference applies to all supported CAD systems and to the DesignModeler application.

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
CAD Associativity (check box)	**Use Associativity** (check box)	Indicates if action should be taken to allow associativity. This option is present because some CAD systems take too long to compute associativity. The default is **Yes**. The preference applies to the following: Autodesk Inventor (p. 40) Associative Geometry Interface CATIA (p. 46) V5 Associative Geometry Interface Creo Elements/Direct Modeling (p. 59) Creo Parametric (p. 62) Associative Geometry Interface (always ON) Fusion (p. 77) Associative Geometry Interface and Reader NX (p. 87) Associative Geometry Interface Solid Edge (p. 101) Associative Geometry Interface SOLIDWORKS (p. 108) Associative Geometry Interface
Coordinate Systems (check box)	**Import Coordinate Systems** (check box)	Specifies whether coordinate systems created in the CAD application should be imported into the AnsysMechanical application. The default is **No**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 35) Reader Autodesk Inventor (p. 40) Associative Geometry Interface CATIA (p. 46) V4 Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader Creo Elements/Direct Modeling (p. 59) Creo Parametric (p. 62) Associative Geometry Interface and Reader NX (p. 87) Associative Geometry Interface and Reader SOLIDWORKS (p. 108) Associative Geometry Interface
Coordinate Systems> Filtering Prefixes	**Coordinate System Key** (text field)	This field can contain any number of prefixes, with each prefix delimited by a semicolon. By default, the filter is empty, which means all coordinate systems are imported.

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
(text field)		
Import Work Points (check box)	**Import Work Points** (check box)	The default is **No**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 35) Reader Autodesk Inventor (p. 40) Associative Geometry Interface CATIA (p. 46) V4 Reader CATIA (p. 46) V5 Associative Geometry Interface and Reader Creo Parametric (p. 62) Associative Geometry Interface Fusion (p. 77) Associative Geometry Interface and Reader NX (p. 87) Associative Geometry Interface Solid Edge (p. 101) Reader SOLIDWORKS (p. 108) Associative Geometry Interface
Reader Save Part File (check box)	**Reader Mode Saves Updated File** (check box)	When set to **Yes**, the interface will save the part file of a model at the end of an update process using the same file name in the same directory. The default is **No**. The preference applies to the following: CATIA (p. 46) V5 Associative Geometry Interface NX (p. 87) Associative Geometry Interface SOLIDWORKS (p. 108) Associative Geometry Interface
Import Using Instances (check box)	**Import Using Instances** (check box)	Processes a CAD model by honoring its part instances to produce faster attach times and smaller database sizes. The default is **Yes**. The preference applies to the following: AutoCAD (p. 34) Associative Geometry Interface Creo Parametric (p. 62) Associative Geometry Interface Fusion (p. 77) Associative Geometry Interface and Reader NX (p. 87) Associative Geometry Interface Parasolid (p. 95) Reader Solid Edge (p. 101) Associative Geometry Interface

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
Smart Update (check box)	**Smart CAD Update** (check box)	Speeds up refresh of models that have unmodified components. If set to **Yes** and changes are made to other preferences, these will not be respected if the component is smart updated. The default is **Yes**. The preference applies to the following: Autodesk Inventor (p. 40) Associative Geometry Interface Creo Parametric (p. 62) Associative Geometry Interface Solid Edge (p. 101) Associative Geometry Interface For more information, see Do Smart Update in the Ansys DesignModeler application help, or the Ansys Mechanical application Help. Note that line bodies are not supported with **Smart CAD Update** and are always marked unmeshed. Any part which has been moved or an instance (p. 133) of it has been moved since last imported will not be refreshed through the Smart Update process. Such parts will always be unmeshed following update.
Compare Parts On Update (drop-down)	**Compare Parts on Update** (drop-down)	For more information, see Compare Parts on Update (p. 22). The default is **No**. The choices are: **No** **Associatively** **Non-Associatively**
Compare Parts Tolerance (drop-down)	**Compare Parts Tolerance** (drop-down)	Use to set the tolerance. The preference is only displayed in the Properties panel if the compare parts have a value to compare. Note that although the choices are the same, the **Compare Parts Tolerance** and **Model Tolerance** in DesignModeler are unrelated. **Tight**: Identifies updated geometry with only very small deviations as unchanged. This is the default. **Normal**: Allows a 100 times greater deviation than **Tight**. **Loose**: Allows a 100 times greater deviation than **Normal**.

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		These deviations are intended to account for modest internal deviations that can sometimes be generated in computational values or in the mathematical representation of some geometry. They are not intended to hide true geometric modifications.
Process Enclosures (check box)	**Enclosure and Symmetry Processing** (check box)	Use to turn on/off the processing of enclosure and symmetry named selections. The default is **Yes**. The preference applies to the following: DesignModeler
Decompose Disjoint Geometry (check box)	**Decompose Disjoint Geometry** (check box)	The default is **Yes**. The preference applies to the following: Creo Parametric (p. 62) Associative Geometry Interface (for use with faces only) Autodesk Inventor (p. 40) Associative Geometry Interface (for use with bodies only) Revit (*.rvt, *.rfa) (p. 98)
Clean Geometry On Import (check box)	**Clean Geometry on Import** (check box)	Use to remove unwanted features when transferring geometry to Ansys Mechanical or Ansys Meshing. Clicking the check box will result in the removal of all unessential edges, vertices, and features; for example, small faces will be merged into larger faces.The default is **No**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 35) Reader CATIA V4 (p. 46) Reader CATIA V5 (p. 49) Reader CATIA V6 (p. 57) Reader Creo Parametric (p. 62) Reader IGES (p. 82) Reader Inventor (p. 40) Reader JT (p. 84) Reader NX (p. 87) Reader Revit (*.rvt, *.rfa) (p. 98) Rhinocerous (p. 99) Reader Solid Edge (p. 101) Reader SOLIDWORKS (p. 108) Reader

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		STEP (p. 116) Reader
Stitch Surfaces On Import (drop-down menu)	**Stitch Surfaces on Import** (drop-down menu)	Use to join surfaces when transferring geometry to Ansys Mechanical or Ansys Meshing. The default is **None**. Note that selection of an IGES file will change the default to **Program Tolerance**. The options are: • **None**: do not stitch surfaces on import. • **Program Tolerance**: stitch surfaces on import using internal algorithm for determining maximum stitch tolerance. • **User Tolerance**: stitch surfaces on import using user-defined **Stitch Surface Tolerance**. The preference applies to the following: ACIS (p. 32) Reader AutoCAD (p. 35) Reader CATIA V4 (p. 46) Reader CATIA V5 (p. 49) Reader Creo Parametric (p. 62) Reader IGES (p. 82) Reader Inventor (p. 40) Reader NX (p. 87) Reader Revit (*.rvt, *.rfa) (p. 98) Rhinocerous (p. 99) Reader Solid Edge (p. 101) Reader SOLIDWORKS (p. 108) Reader STEP (p. 116) Reader
Stitch Surface Tolerance (value field)	**Stitch Surface Tolerance** (value field)	This field is displayed if the **Stitch Surfaces On Import** option is set to **User Tolerance**. The field contains the user-defined stitch tolerance. The Stitch Surface Tolerance is always in meters. By default, the tolerance is 1E-07.
Mixed Import Resolution (drop-down)	**Mixed Import Resolution** (drop-down)	Allows parts of mixed dimension to be imported as components of assemblies which have parts of different dimension. The options are: **None**: If there are bodies of mixed dimension in a multibody part, nothing is transferred to the Ansys Mechanical application

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		Solid: Only solid(s) from the part are transferred to the Ansys Mechanical application **Surface**: Only surface(s) from the part are transferred to the Ansys Mechanical application **Line**: Only line(s) from the part are transferred to the Ansys Mechanical application. **Solid and Surface**: Only solid(s) and surface(s) from the part are transferred to the Ansys Mechanical application. **Solid and Line**: Only solid(s) and line(s) from the part are transferred to the Ansys Mechanical application. Option is not supported for SOLIDWORKS Associative Geometry Interface. **Solid, Surface, and Line**: Solid(s), surface(s), and line(s) from the part are transferred to the Ansys Mechanical application. Option is not supported for SOLIDWORKS Associative Geometry Interface. **Surface and Line**: Only surface(s) and line(s) from the part are transferred to the Ansys Mechanical application. This option is controlled by the feature in DesignModeler. See Expected Body Imports (p. 21) for more information.
Import Facet Quality	**Import Facet Quality**	The options listed below are available for import facet quality. The default value is Source meaning that the facets as represented in the CAD system or use a Normal option for those that do not have display. The others are relative settings compared to "normal" facet quality. Typically, better facet quality requires more memory and may take additional time to import/update. Very Coarse Coarse Normal Fine Very Fine Source The preference applies to the following: AutoCAD (p. 34) Associative Geometry Interface Creo Parametric (p. 62) Associative Geometry Interface

Workbench Options> Geometry Import Selections	Project Schematic Selections	Description
		Fusion (p. 77) Associative Geometry Interface and Reader IGES (*.igs, *.iges) (p. 82) Autodesk Inventor (p. 40) Associative Geometry Interface NX (p. 87) Associative Geometry Interface Revit (*.rvt, *.rfa) (p. 98) STEP (*.stp, *.step) (p. 116)
Post-Import Script (**Browse** button to **Open** file dialog)	**Post-Import Script** (**Browse** button to **Open** file dialog)	A script that will be executed after the completion of the attach of an active geometry when using the Workbench button in the Ansys menu inside the CAD system. This extensible functionality allows you to create and import into a specified system, set geometry import preferences for the import, and execute other scriptable functionality. Simple example scripts can be found in *<Ansys_InstallDir>*\aisol\CommonFiles\CommonPages\PostImportScripts.

Expected Body Imports

The following table shows the expected body imports based on the composition of the part (top row) and the mixed dimension import resolution preference. It is assumed for this table that the body types indicated in the part are selected in the primary import options.

	Solid-Surface-Line	Solid-Surface	Solid-Line	Surface-Line
None	No import	No import	No import	No import
Solid	Solid	Solid	Solid	No import
Surface	Surface	Surface	No import	Surface
Line	Line	No import	Line	Line
Solid and Surface	Solid and Surface	Solid and Surface	Solid	Surface
Solid and Line	Solid and Line	Solid	Solid and Line	Line
Surface and Line	Surface and Line	Surface	Line	Surface and Line
Solid, Surface and Line	Solid, Surface and Line	Solid and Surface	Solid and Line	Surface and Line

This processing becomes significant after handling the basic import options (for example, if a part is Solid-Surface-Line, if only Import Solids is selected, then only solid bodies would be imported) regardless of the Mixed Import Resolution (p. 24).

However, when transferring the model from the Ansys DesignModeler or SpaceClaim application to the Ansys Mechanical application for a 3D Analysis, the basic import options and Mixed Import Resolution (p. 24) option are not applicable. All unsuppressed bodies/parts in the model will be transferred to the Ansys Mechanical application.

Note:

The Solid and Line option and the Solid, Surface and Line option are not supported for SOLIDWORKS Associative Geometry Interface.

Note:

In the case of a 2D analysis, you should suppress all solid bodies in DesignModeler for a successful transfer.

Attributes, Coordinate System, Named Selection, and Parameter Filter Usage

Default filters that are inclusion filters (for example, NS, ANS;DS, and SDFEA;DDM) are available. These indicate that if the required object includes that string at the beginning (or end for parameters) that the entity's information will be filtered in and equivalent data will be created. If the filter is removed entirely it is assumed that all data is to be processed.

Ansys also supports a negation filter (a string following a -), to filter data out. For example, a parameter filter of DS;-DS_N would import parameters that start with DS, but do not start with DS_N. The negation filter can also be used independently, the key thing being that the inclusion filter is assumed to include all data. By example, a parameter filter of -STRUCTURAL would import all parameters of a model except those that include the string STRUCTURAL at the beginning or end of their name.

Compare Parts on Update

Compare Parts on Update is a **Geometry Import** option, available inside Ansys Workbench. The Ansys Mechanical editor and Ansys Meshing make use of the benefits of the preference.

The option compares the existing geometry and updated-to geometry to determine if any bodies/parts remain unmodified. When you choose the **Refresh Geometry** context menu option (right-click the **Geometry** tree object or anywhere in the Geometry window), if no changes to the body are detected (found to be unmodified), then the body/part does not need to be remeshed after the update has occurred.

The option can be set or modified before any update in the Ansys Mechanical editor or Ansys Meshing and the subsequent updates will utilize that option.

For more information, see **Compare Parts on Update** in the Advanced Geometry Options table at Geometry Preferences (p. 8) in the **CAD Integration** section of the Ansys Help.

Note:

The Non-Associative setting for **Compare Parts on Update** depends on the entities from the updated model being in the same order as the original model. If that order changes at any level, that part will be marked as modified. For most CAD systems if the CAD session remains open the order is more likely maintained then if the CAD is shut down and the model reopened. For those geometry imports that support associativity the associative option for **Compare Parts on Update** is the preferred mechanism.

The **Compare Parts on Update** feature is not supported for line bodies.

If the final existing instance of a part is modified such that it is no longer instanced, an associative **Compare Parts on Update** may result in both the un-modified and the modified part being marked as un-meshed, due to a change in the transformation matrix of the un-modified part.

Accessibility

From the **Tools** menu in Ansys Workbench, select **Options** to display a dialog box and then, select **Geometry Import**. The **Advanced Options** category includes the **Compare Parts on Update** option.

- The choices are:
  - **No** (default): No post-update comparison will be attempted and all bodies/parts included in the update will be marked as dirty and needing remeshed.
  - **Associatively**: Searches through the part data to find appropriately matching entities for comparison based on their associative data. This option is slower because of that search but provides the best option if parts or bodies are reorganized or added to or deleted from. The **Associatively** option is the recommended comparison option and should be used exclusively if the model utilizes merged topology from the DesignModeler application.
  - **Non-Associatively**: Generates a faster comparison but only does comparison on entities of a corresponding index (body 1 original -> body 1 updated). If bodies are not reorganized or added to or deleted from this method should be adequate, as long as the original topology is preserved. You should not use the **Non-Associatively** option when using merged topology options (Automatic or Imprints) from the DesignModeler application.

The three options for Tolerance are:

- **Tight** (default): the Tight setting identifies updates geometry with only very small deviations as unchanged.
- **Normal**: the Normal setting allows a 100 times greater deviation than tight.
- **Loose**: the Loose setting is 100 times greater than normal.

These deviations are intended to account for modest internal deviations that can sometimes be generated in computational values or in the mathematical representation of some geometry. It is not intended to hide true geometric modifications.

Mixed Import Resolution

Mixed-dimension parts are parts that contain bodies of differing dimension (solids and surfaces, surfaces and lines, solids and lines, or solids, surfaces, and lines). When importing mixed-dimension parts, the Mixed Import Resolution preference controls which dimensional sets of these bodies are imported. For example, if the part contains solids, surfaces, and lines and the Mixed Import Resolution preference is set to Surface, only the surface bodies contained in the part will be imported. The option has no effect on the import of single-dimension parts.

For more information about Mixed Import Resolution, see the tables in Geometry Preferences (p. 8).

Basic Options:Basic Geometry Options (p. 9)

Advanced Options:Advanced Geometry Options (p. 14)

Product & CAD Configuration Manager

The Product & CAD Configuration Manager allows you to configure geometry interfaces for:

Mechanical APDL (Ansys)

Ansys Workbench

CAD configuration is typically handled during the product installation. However, if you chose to skip those steps, or if you make changes to your local CAD configuration between releases (for example, you install, move, or update your CAD package, or remove it entirely), you can use this utility.

For platform specific information to use the Product & CAD Configuration Manager, see:

- Ansys, Inc. Windows Installation Guide> Configuring CAD Products> Using the CAD Configuration Manager
- Ansys, Inc. Linux Installation Guide> Configuring CAD Products> Using the CAD Configuration Manager on Linux

Topics covered include:

- Unconfiguring
- Running the Product & CAD Configuration Manager in Batch Mode
- Uninstalling
- Creo Parametric Configuration
- NX Configuration
- Configuring CADNexus/CAPRI CAE Gateway for CATIA V5

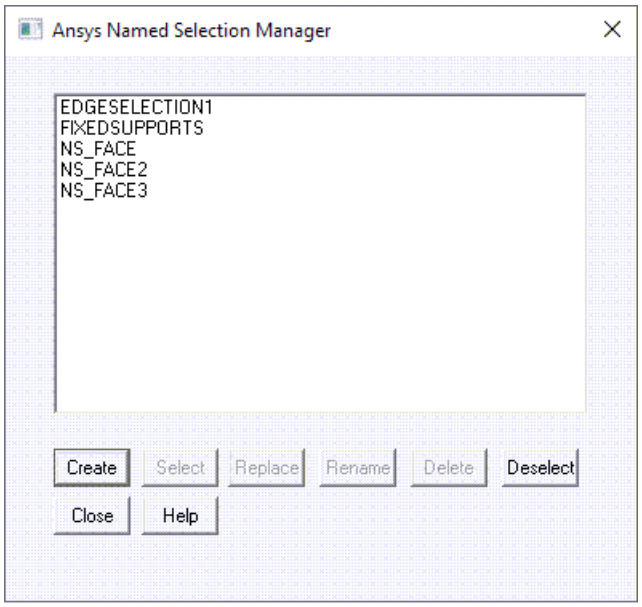
Notations

If the help for the Product & CAD Configuration Manager does not load into your default browser, set the BROWSER environment variable to the path of your HTML viewer (such as Mozilla or Firefox) and restart the Product & CAD Configuration Manager.

Named Selection Manager

Use the **Ansys Named Selection Manager** to create custom attributes with CAD systems for modeling, meshing and performing analysis within Ansys Workbench.

This mechanism is activated by selecting the **Named Selection Manager** menu item from the current Ansys release menu inside the CAD system (with the exception of Creo Elements/Direct Modeling (p. 59) and CATIA (p. 46)).



The **Named Selection Manager** displays groups created only in this tool but the Plug-in (p. 133) will continue to support those previously imported in earlier Ansys releases. The Named Selections are always listed alphabetically. The **Named Selection Manager** supports defining Name Selection Groups within CAD through the following operations.

- **Create**: Pre-select entities to designate for the NS Group, click "Create" button and designate a name for the group. Some CAD systems (for example Solid Edge (p. 101) and NX (p. 87)) allow the selection between the Create and Naming operations.
- **Delete**: Select a group from the **Named Selection Manager** list followed by clicking the "Delete" button. The Delete option allows you to delete the chosen named selection. Note that a confirmation dialog box does not appear before that deletion.
- **Rename**: Select an entry from the **Named Selection Manager** group list, click "Rename" and supply a new group name. The Rename option allows you to change the name of the selected group.
- **Replace**: Pre-select entities to compose the Named Selection, click "Replace". The newly selected set will replace the entities in the Named Selections. No previous entities will remain except those which were selected at the time of the "Replace" operation.
- **Select/Deselect**: Help tools which allow for verification of a Named Selection Group's contents.
- **Close**: Closes the existing **Named Selection Manager** dialog.
- **Help**: Opens Ansys Workbench help.

Note that in the case of NX (p. 87), the geometry selection dialog box will prompt you when to click the **Select** option. To modify a named selection, select the entry in the list and click the **Replace** option. The geometry selection dialog box will pop up again to let you modify the selection created earlier.

Named Selection Import based on Entities

A check mark represents entities that are supported by the CAD system in the first column.

CAD System		Vertex	Edge	Face	Body
Autodesk Inventor (p. 40)	Part Level	√	√	√	
	Assembly Level	√	√	√	
DesignModeler		√	√	√	√
CATIA (p. 46) V5	Part Level	√	√	√	√
	Assembly Level	√	√	√	√
Creo Parametric (p. 62)	Part Level		√	√	√*
	Assembly Level		√	√	√*
Solid Edge (p. 101)	Part Level	√	√	√	
	Assembly Level				
SOLIDWORKS (p. 108)	Part Level	√	√	√	√
	Assembly Level	√	√	√	
NX (p. 87)	Part Level		√	√	√
	Assembly Level		√	√	√

* Supported for Creo 7 and 8 (by implication not 6).

GUI Navigation: Creating Named Selections in CAD System to Import into Workbench

To create named selections within a CAD system for importation into Ansys Workbench:

1. From an active CAD session, select the element (for example faces) of the model to which you want to apply a named selection to the **Named Selection Manager**.
2. Select **Named Selection Manager** from the Ansys drop-down menu to display a dialog box.
3. Select Create to display a second dialog box (Name Input Dialog).
4. Enter a name for the selected element. Select OK to list the name in the Ansys Named Selection dialog box.
5. Highlight the Named Selection to activate the Select, Replace, Rename, and Delete options in the dialog box. Choose an option, and select Close.
6. Import the active model into the desired application making sure to set the Named Selections preference on and the Named Selections Key matching the desired sets for import.
7. Select Workbench from the Ansys drop-down menu to start an Ansys Workbench session.

8. The CAD system icon in the Geometry cell of the Project Schematic indicates the model is loaded. Right-click the Geometry cell to select Edit, or double-click cell.

9. Choose desired length in the Units dialog box, and select OK. An object named "Attach1" appears in the tree outline.

10. In the Details View, select Yes to Import Named Selections.

11. In the Details View, the Named Selection Key must match the CAD system prefix.

12. Select Generate on the 3D features toolbar to complete the Attach feature. The Named Selection created in the CAD system appears under the "Attach1" object in the tree outline.

Caveats and Known Issues

Geometry Import Fails

Ansys Workbench leverages a running CAD session to update models if one exists. If you change the geometry source to a file that cannot be opened in the running session, the update will fail with an unable to activate document error. Some reasons this may occur are if the new model is from a version of the CAD system later than the one running or if another file of the same name is already opened in the CAD from another directory.

Length Units When Changing Geometry Sources

The length units of loads and boundary conditions are based on the unit of the imported model in Mechanical. When the source geometry is changed to another model which uses a different length unit than the first, then loads, boundary conditions, and non-associative coordinate systems in Mechanical will scale to match the new length unit of the imported geometry. This may be observed when importing a model into Mechanical, then later choosing to edit the geometry in DesignModeler. If the DesignModeler session's length unit is different than the original CAD model's length unit, then any existing loads or boundary conditions in Mechanical that were defined for the original CAD model may get scaled.

CAD Integration Installation and Licensing

Because the individual CAD systems are listed with the other components during the product installation, you can select the CAD interfaces that you wish to install.

Some product components, including some CAD interfaces, require administrative permissions to register. If you install with non-administrative privileges, you must follow the post-installation procedures as an administrator for your product to ensure that all components are successfully registered.

For CAD-related installation prerequisites, see Table 1–3 (Product Support) in the following sections:

- Installation Prerequisites for Windows
- Installation Prerequisites for Linux

For CAD-related installation information, see:

- Ansys, Inc. Windows Installation Guide> Configuring CAD Products
- Ansys, Inc. Linux Installation Guide> Configuring CAD Products

For licensing information, see the **Product to License Feature Mapping** table located on the Ansys Customer Portal. To access the table, log on to the Ansys Customer Portal, click **Downloads> Installation and Licensing Help and Tutorials** and expand the **Licensing** section of the page.

CAD Integration File Format Support

Within Ansys Workbench, the CAD files can be attached in either Plug-In or Reader mode.

Plug-In mode: requires that the CAD system be running.

Reader mode: does not require the CAD system to be running.

Pseudo-Reader mode: CAD system is started by Ansys Workbench in batch mode and shuts it down after attach/update is completed.

File Format Supported:

- ACIS (*.sat, *.sab) (p. 32)
- Ansys Part Manager (*.pmdb) (p. 34)
- AutoCAD (*.dwg, *.dxf) (p. 34)
- Autodesk Inventor (*.ipt, *.iam) (p. 40)
- CATIA (*.model, *.exp, *.session, *.CATPart, *.CATProduct, *.3dxml) (p. 46)
- **CoCreate Modeling**: see Creo Elements/Direct Modeling
- Creo Elements/Direct Modeling (*.pkg, *.bdl, *.ses, *.sda, *.sdp, *.sdac, *.sdpc) (p. 59)
- Creo Parametric (*.prt, *.asm) (p. 62)
- DesignModeler (*.agdb) (p. 76)
- Discovery (*.dsco) (p. 80)
- Fusion (*.f3d, *.f3z) (p. 80)
- IGES (*.igs, *.iges) (p. 82)
- JT Reader (*.jt) (p. 84)
- Monte Carlo N-Particle (*.mcnp) (p. 86)
- NX (*.prt) (p. 87)
- **OneSpace Designer Modeling**: see Creo Elements/Direct Modeling
- Parasolid (*.x_t, *.xmt_txt, *.x_b, *.xmt_bin) (p. 95)
- **Pro/ENGINEER**: see Creo Parametric
- Revit (*.rvt, *.rfa) (p. 98)

- Rhinoceros (*.3dm) (p. 99)
- Solid Edge (*.par, *.asm, *.psm, *.pwd) (p. 101)
- SOLIDWORKS (*.sldprt, *.sldasm) (p. 108)
- SpaceClaim (*.scdoc, *.scdocx) (p. 114)
- STEP (*.stp, *.step) (p. 116)

ACIS (*.sat, *.sab)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the ACIS system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface is configured during Ansys installation by default.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Linux: Configuring CAD Products> Using the CAD Configuration Manager

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.sat,*.sab`)

Versions: R1 — 2023 (Windows & Linux)

Table 1: Import Preference Support for ACIS Geometry Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color, Layer, and Publication
Named Selection Processing and prefix	Yes - Color, Layer, and Publication

Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Assembly

Although the ACIS geometry format does not have an assembly entity, the application supports ACIS files containing one or multiple bodies.

Hidden Parts

Parts that are hidden or suppressed in ACIS are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").

- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Import CAD Publication as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Publication" or any number of prefixes with each prefix delimited by a semicolon (for example: "NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Publication" or any number of prefixes with each prefix delimited by a semicolon (for example: "NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded").

Caveats and Known Issues

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

See the ACIS Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

Ansys Part Manager (*.pmdb)

Notations

Import Preferences

When importing Ansys Part Manager files, import preferences will not be shown. This is because these files are essentially a snapshot of a prior import. They will simply resume the geometry data exactly as it was originally imported.

Faceted Geometry

DesignModeler does not support the import of Ansys Part Manager files in which virtual operations in Mechanical were applied to the geometry prior to exporting the *.pmdb file.

AutoCAD (*.dwg, *.dxf)

For more information, see:

AutoCAD Reader (*.dwg, *.dxf)

AutoCAD Associative Geometry Interface (*.dwg, *.dxf)

AutoCAD Reader (*.dwg, *.dxf)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the AutoCAD system be installed.
- No CAD associativity support.
- No CAD Parameter support.

The interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.dwg`, `*.dxf`

Versions: 2.5 - 2019

Table 2: Import Preference Support for AutoCAD Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color and Layer
Named Selection Processing and prefix	Yes - Color and Layer
Material Processing	No
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	No

Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden Parts

Parts that are hidden or suppressed in AutoCAD are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Caveats and Known Issues

The AutoCAD Reader has the following limitations:

- **No AEC and AECB entries.**
- **No support for paper spaces.**
- **No support for block properties (Color, Layer).**
- **No support for Insert Layer Mode color handling.**

- **No support for PMI & Text.**
- **No support for Body names.**

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

See CAD Integration Troubleshooting (p. 127) for detailed information.

AutoCAD Associative Geometry Interface (*.dwg, *.dxf)

The interface works in both Plug-in (p. 133) and Pseudo-Reader (p. 133) mode.

See the Product & CAD Configuration Manager (p. 24) for usage information.

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.dwg`, `*.dxf`

Table 3: Import Preference Support for AutoCAD Geometry Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	No
Named Selection Processing and prefix	Yes – AutoCAD Layers can be imported as Named Selections
Material Processing	No
Analysis Type	3D - Yes 2D - Yes; surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No

Reader Save File	No
Instancing	Yes
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes: for parts that include more than one body type (solid, surface, line) with these selected for import.
Decompose Disjoint Geometry	No
Import Facet Quality	Yes

Note:

AutoCAD LT is not supported. You must have the full version of the product to perform an import.

Notations

Entities which belong to layers which are invisible, frozen or turned off will be skipped during import.

AutoCAD mesh bodies, of type Polygon Mesh and Polyface Mesh, are not supported for import. Note that when exporting Revit models to DWG the default solid type is Polyface. To generate a DWG file with Workbench importable bodies, it is required that the Revit user select "ACIS Solids" in the Solids tab of the DWG/DFX Export Setup dialog.

Body Arrays

Geometry created by using the AutoCAD "Array" command will not automatically be imported into Ansys Workbench. To make such bodies importable: (a) load the drawing file into AutoCAD, (b) select the array, and (c) issue the "Explode" command.

Color Processing

AutoCAD bodies can be automatically grouped in Named Selections based upon their color by adding the keyword "Color" to the Named Selection Key list. The resultant selection groups will be named with the form, Color:R.G.B. Here R, G and B are values ranging from 0 to 255, which represent the amount of color saturation of Red, Green and Blue color components. For example Color:0.0.0 is black, Color:255.255.255 is white, and Color:255.0.0 is red.

Within the Mechanical application, body colors can be visually altered to match what is displayed in the AutoCAD session by executing the following steps:

1. Prior to attach/refresh, enable the Attributes preference.
2. Add the keyword "Color" to the Attributes Key list.
3. After attach/refresh is complete, within Mechanical perform Tools->Run Macro and designate this script to be run: `<install dir>\v<version>\aisol\DesignSpace\DSPages\macros\BodyColorByCADColorAttribute.js`

Layer Processing

AutoCAD entities can be automatically assigned to Named Selections based upon their layer assignment. This can be accomplished by enabling the Named Selection preference, and adding the keyword "Layer" to the Named Selection Key list. Subsequent attach/refresh actions will import with selection groups named with the form, Layer:<*AutoCAD Layer Name*>.

Localization Considerations

To change the Ansys Ribbon to match Workbench localization follow these steps:

1. Unconfigure the AutoCAD Plug-In using Product & CAD Configuration Manager (p. 24)
2. Run AutoCAD one time, then close.
3. Under the options menu, change locale setting within Workbench.
4. Configure the AutoCAD Plug-In using Product & CAD Configuration Manager (p. 24).
5. Run AutoCAD.

Pseudo-Reader Attach/Refresh

When importing and no active session of AutoCAD is running, Ansys Workbench will automatically launch the last run version of AutoCAD for the current user account. Import will fail if the current user has never launched AutoCAD. In this case launching the CAD system one time will resolve the failure. When DWG TrueView is present and has been run prior to the most recent execution of AutoCAD, pseudo-reader attach will fail. In this case, launching AutoCAD one time will correct the problem.

Uninstall Considerations

When the AutoCAD associative interface is configured for a non-admin user the Ansys ribbon tab will remain after the product is uninstalled. To avoid this the non administrative user should manually unconfigure the product prior to removing the installation, using the Product & CAD Configuration Manager (p. 24).

Units

ACAD model units of Meters, Inches, Feet, Millimeters, Centimeters, Microns, and Nanometers will be transferred as the specified model unit. All other unit settings will be marked as unknown unit type (as had been the case for all ACAD imports previously).

Limitations

There is a known when launching Workbench directly from AutoCAD, attempts to import the AutoCAD model into Mechanical will result in an error opening that application. The workaround is to launch Workbench from the start menu and link the active AutoCAD model to the Geometry cell using the right mouse button and Import Geometry option.

The following limitation is specific to the Ansys 2025 R1 and 2024 R2 Associative Geometry Interfaces to AutoCAD. There is a known library conflict with earlier releases of this Plug-In, which causes the 2025 R1 Ansys Plug-In to fail loading into AutoCAD when a previous version of the Plug-In is configured to load into AutoCAD.

Use the following procedure to enable these versions to be loaded side-by-side in AutoCAD.

1. Use Product & CAD Configuration Manager (p. 24) to configure the AutoCAD Associative Geometry Interface for 2025 R1.
2. Unconfigure all other versions of the AutoCAD Associative interface, also using the corresponding version of Product & CAD Configuration Manager.
3. Open AutoCAD, the 2025 R1 version of the Ansys Geometry Interface will automatically load. All other versions will need to be manually started by:
   - Start the Application
     - Within AutoCAD open a model such as a DWG or DFX file.
     - Manage >>Load Application
     - Browse to <Path to Ansys Installation>\aisol\CADIntegration\DWG\winx64
     - Select WB<version>PlugInDWG2021.arx and click Load
   - Load the Robbon Tab
     - Manage >> User Interface
     - Select Partial Customization Files in the tree, and Right Mouse Button >> Load Partial Customization File
     - Browse to <Path to Ansys Installation>\aisol\CADIntegration\DWG\Language\<language>
     - Open Workbench241.cuix
   - Note for future startup of AutoCAD step (b) will not be needed; however, you must complete the steps to start the application enumerated under (a).

Autodesk Inventor (*.ipt, *.iam)

For more information, see:

Autodesk Inventor Reader (*.ipt, *.iam)

Autodesk Inventor Associative Geometry Interface (*.ipt, *.iam)

Autodesk Inventor Reader (*.ipt, *.iam)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the Autodesk Inventor system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.ipt`) and Assembly (`*.iam`)

Versions: .ipt (6 — 2025), .iam (11 — 2025)

Table 4: Import Preference Support for Autodesk Inventor Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes – Color
Named Selection Processing and prefix	Yes – Color
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- Only surface bodies in the xy plane will be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden Parts

Parts that are hidden or suppressed in Autodesk Inventor are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Caveats and Known Issues

The Inventor Reader has the following limitations:

Limited support for Inventor Assembly.

- Assembly attributes such as colors and layers are not supported.
- Assembly level features are not supported. For example, an instance can be marked as suppressed (that is, not visible) in an Inventor assembly. Because the reader does not support reading suppressed information, suppressed instances are translated.

No Support for Sketches.

The reader currently does not support sketches from Inventor files.

Limited Entity Support for Inventor.

The reader currently does not support some specific entities resulting from advanced feature Inventor operations such as Lofting.

No Support for Layers.

The reader does not support translating layer information.

No Support for Welding Symbols.

The reader does not support Weld symbols from Inventor Files.

BREP Color Limitation.

Body colors are not supported for parts having multiple bodies with the same number of faces. No color support for files created in a local, non-English Inventor Modeler.

Material Properties Limitation.

- Only isotopic material properties and metric standard units (MPa, Kg, J) are supported.
- Currently material properties for default Inventor material (that is, "Generic" shown in Inventor Modeler) are not supported.
- The reader translates material properties for Inventor version 2012 and later.

No Support for Free Form Surface.

The translator does not support Free Form Surface (T-Splines).

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

No known issues have been identified in the CAD Integration Troubleshooting (p. 127) section.

Autodesk Inventor Associative Geometry Interface (*.ipt, *.iam)

The interface works in both a Plug-in (p. 133) and a Pseudo-Reader (p. 133) mode.

The existence of the Plug-in (p. 133) is recognized by Autodesk Inventor based on registry entries. If the Plug-in (p. 133) is not available in the add-in manager in Autodesk Inventor you will need to run the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Table 5: Import Preference Support for Autodesk Inventor Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	No
Parameter Processing and prefix/suffix key	Yes

Attribute Processing and prefix	Yes - including Color
Named Selection Processing and prefix	Yes - including Color Yes - from Ansys Named Selection Manager (p. 25) and programmatically created attributes
Material Processing	Yes
Analysis Type	3D - Yes 2D - Surface bodies in only the xy plane will be imported
Associativity	Yes
Coordinate Systems	Yes - part and assembly user coordinate systems
Work Points	Yes
Reader Save File	No
Instancing	Yes
Smart Update	Yes
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes - for parts that include both Solid and surface bodies AND the Import Solid and Import Surface preferences are set to Yes
Decompose Disjoint Geometry	Yes (for use with bodies only)
Import Facet Quality	Yes

Notations

The Autodesk Inventor geometry interface also supports the Selective Update feature.

Assembly parts and sub-assemblies that are hidden, suppressed or disabled in Autodesk Inventor are automatically excluded from import.

FaceColor attributes in Autodesk Inventor will now be separated based on their assigned value when imported as named selections. A model that may have 3 Face entities assigned red and 2 Face entities that were assigned blue which previously would have imported as a single named selection titled FaceColor, including all five faces, will now import as two named selections, FaceColor:Red (3 faces) and FaceColor:Blue (2 faces).

Ansys Workbench automatically locks the length unit in the part or assembly to centimeters, which is the unit used internally by Autodesk Inventor. No adjustment of length unit is necessary or possible. The Ansys Mechanical application user can change the unit system for display of the Ansys Mechanical application data.

Specific to Ansys DesignModeler

To successfully import parts into the Ansys DesignModeler application, you may need to suppress the elliptical features from the tree in Autodesk Inventor.

Large models (up to 499999m from the origin in any direction) created in Autodesk Inventor can be imported to the DesignModeler application. Note that the DesignModeler application reports the bounding box of 500 km for such entities, that is the tolerance for Autodesk Inventor is 1 m.

Autodesk indicates that Shrinkwrap of Inventor native assemblies can produce non manifold bodies. In this case the Inventor body can display normally in the CAD's model tree, but is not expected to import cleanly into DesignModeler due to gaps where small faces were removed.

Bodies imported into Inventor from external sources can produce non-manifold solids, which load into DesignModeler with less than okay status and missing faces. Right-clicking on these bodies within Inventor will provide access to the Body Repair tools offered within the CAD. These utilities can be used to identify and heal errors in the Inventor geometry, which in turn can improve translation into DesignModeler.

When transferring geometry from Autodesk Inventor to the Workbench using the associative geometry interface, a specific representation is not loaded into Autodesk Inventor when opening a document via Pseudo-Reader (p. 133) mode. If a specific representation is required, it should be explicitly set in an active Autodesk Inventor session prior to attach.

Caveats and Known Issues

DSPlugInAICOM Add-in

The first time you start Inventor after installing/configuring the plug-in you may be presented with a dialog box asking whether you would like to load the DSPlugInAICOM add-in. Here if you click Allow the Associative Interface will automatically load in the future without the message box appearing again.

Decompose Disjoint Geometry

Each component shell will be imported as a distinct body. Decomposition is only supported for solid bodies, and disjoint surface bodies such as face patches and work surfaces will not be broken apart.

Import and Mesh Efficiency

To improve import and mesh efficiency the import or part instancing. is supported. As a result, a break in associativity is expected when performing an update for models from versions of Workbench earlier than 16.0. The work-around is to disable the Instancing preference prior to refreshing the geometry from CAD, with the downside of losing out on the performance improvements yielded via Instancing.

Model States Considerations

Any update from an Inventor Model State different from the version currently loaded into a downstream Ansys application such as Mechanical, Discovery, SpaceClaim, DesignModeler, or Fluent has potential to result in broken loads, boundary conditions or other simulation entities assigned to a specific topological entity or part. This can be mitigated by defining Named Selections in the Inventor model and scoping simulation items to these Named Selections. For more information regarding creation of Named Selection see documentation about using the Named Selection Manager (p. 25).

Named Selections Restrictions

The Autodesk Inventor Plug-in (p. 133) does not support the creation of Named Selections beginning with numeric characters, containing spaces, the slash (/) or backslash (\) characters.

Plug-In Availability Considerations

Due to the architecture of the CAD's Add-In manager, the Autodesk Inventor geometry interface will automatically load into the Inventor sessions of all users when any administrative user has configured the geometry interface only to be available for their account. This is the case even when you have not configured the geometry interface for himself/herself and a global configuration has not been performed to enable the plug-in to run for all users. When encountering such a state it may be possible to import geometry from an active Inventor session, but any attempts to attach or refresh geometry without Inventor running will fail.

Reference Key

If, upon attaching, you receive the message *Failed to get reference key*, the attaching process will continue, but an associative relation during update cannot be guaranteed.

Limitations

The following limitation is specific to the Ansys 2025 R1 and 2024 R2 Associative Geometry Interfaces to Inventor. There is a known library conflict with earlier releases of this Plug-In, which will cause Inventor to crash when attempting to load 2025 R1 Ansys Plug-in when earlier release of the plugin is configured to load into Inventor.

Use the following procedure to enable these versions to be loaded side-by-side in Inventor.

1. Open CAD Configuration Manager 2025 R1 and unconfigure the Inventor Associative Geometry Interface
2. Configure all other desired versions of the Inventor Geometry Interface, using the corresponding version of CAD Configuration Manager
3. Open Inventor. Tools >> Add-Ins. For each Workbench version: Uncheck – Load Automatically. Uncheck – Block.
4. Use 2025 R1 CAD Configuration Manager to configure the Inventor Associative Interface.
5. Open Inventor. Tools >> Add-Ins. Select Workbench 2025 R1 and Check – Load Automatically. Uncheck – Block.
6. Each time Inventor is started you will need to open the Inventor Add-In manager and check Loaded for Workbench versions earlier than 2025 R1.

Troubleshooting

See the Autodesk Inventor Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

CATIA (*.model, *.exp, *.session, *.CATPart, *.CATProduct, *.3dxml)

For more information, see:

CATIA V4 Reader (*.model, *.exp, *.session)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone Reader (p. 133) which does not require that the CATIA V4 system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.model`, `*.exp`, `*.session`)

Versions: 4.1.9 — 4.2.5

Table 6: Import Preference Support for CATIA V4 Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color and Layer
Named Selection Processing and prefix	Yes - Color and Layer
Material Processing	No
Analysis Type	3D - Yes

	2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden parts

Parts that are hidden or suppressed in CATIA are skipped automatically by the geometry interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Caveats and Known Issues

The CATIA V4 Reader has the following limitations:

Unsupported CATIA file formats

Does not provide the functionality to read CATIA assembly (*.asm) and CATIA drawing files (*.drw).

Model history information

Does not provide the functionality to read the modeling history maintained in CATIA.

VIEW or DITTO layers

A layer belonging to VIEW or DITTO is not read. The layer filters for VIEW and DITTO are different and are not supported. The layer 255 is meant only for DITTO, which is not supported.

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

See the CATIA Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

CATIA V5 Reader (*.CATPart, *.CATProduct)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the CATIA V5 system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Linux: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) and Linux (p. 4) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.CATPart`) and Assembly (`*.CATProduct`)

Versions: V5R8 – V5–6R2024 (Windows & Linux)

Table 7: Import Preference Support for CATIA V5 Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color, Layer, and Publication
Named Selection Processing and prefix	Yes - Color, Layer, and Publication
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies in the xy plane may be imported
Associativity	No
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update (Windows)

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden parts

Parts that are hidden or suppressed in CATIA V5 are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Import CAD Publication as an Attribute or Named Selection (Windows)

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Publication" or any number of prefixes with each prefix delimited by a semicolon (for example: "NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Publication" or any number of prefixes with each prefix delimited by a semicolon (for example: "NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded").

Caveats and Known Issues

The CATIA V5 Reader has the following limitations:

CATIA V5 R8 and earlier files are not supported

CATPart files created in V5R8 or earlier that have been opened and re-saved in a later version of CATIA V5 are not supported.

How to convert CATIA V5 R8 and earlier files to a newer supported version which will import with CATIA V5 Reader.

1. open original CATPart and copy PartBody
2. open new Part then “Paste special” with “as result” option in part
3. close original CATPart
4. save new CATPart

File Names.

The reader supports reading file names only within the ISO-646 subset of characters with the following limitations (file names with the following special characters are not supported):

- > (greater than)
- < (less than)
- * (asterisk)
- : (colon)
- " (quotation mark)
- ? (question mark)
- \ (backslash)
- | (vertical bar)
- / (slash)
- Additionally, all the *national accented characters* are also not supported.
- The Reader reads file names having their full path. The reader cannot read the file by file name only , or by using a relative path (for example, "...\", "\\"or ".\").

Reading Manufacturing Information.

The reader does not support reading:

- Hole features from assembly files
- User-defined patterns and its member hole features
- Product Manufacturing Information without geometry ownership

No Support for Transparency from an Assembly File.

The reader does not support reading transparency values in assembly (CATProduct) files.

No Support for Captures from an Assembly File.

The reader does not support reading captures from assembly (CATProduct) files. Currently, support for captures is limited to part (CATPart) files only.

No Support for CATPart Files Created in Small or Big Scale.

The CATIA V5 UI allows CATPart files to be created in Small or Big Scale. Also, the CATIA V5 UI allows the creation of CATProduct files referring to CATPart files from different Scales. The CATIA V5 Reader does not support reading of CATPart files created in Small or Big Scales. Similarly for the same reason, CATProduct files referring to such CATPart files might not be read partially or completely.

Publication

Publication name defined with English characters is supported. Publication name defined with non-English characters is not supported.

Small Scale

Small Scale models require the CATIA V5 Geometry Scale to be set to Small Scale. To change the scale setting to allow the import of Small Scale models:

1. Open a native CATIA V5 session
2. Select Tools: Options
3. Select General: Parameters and Measure
4. Select Scale tab
5. Change Geometry Scale from "Standard Scale" to "Small Scale"
6. Exit native CATIA V5 session
7. Set environment variable: `CATUserSettingPath=C:\Users\MyAccount\AppData\Roaming\DassaultSystemes\CATSettings`
8. Now start Ansys Workbench and the CATIA V5 model will import correctly. Note: replace "C:\Users\MyAccount\AppData\Roaming" with your %appdata% value

Body Naming

Requires libXm.so (included in Motif package).

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

See the CATIA Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

CATIA V5 Associative Geometry Interface (*.CATPart, *.CATProduct)

The interface works in Pseudo-Reader (p. 133) mode.

The interface can be configured during Ansys installation if the prerequisites are installed, or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Prerequisites

The CATIA V5 associative geometry interface requires that the following software be installed and running:

- Compatible version of CATIA V5 CAD program (V5–6R2022, V5–6R2023, V5-6R2024).
- DSLS configured with a CATIA V5 (MD2, HD2, or ME2) license.
- CADNexus CAPRI CAE Gateway V4.10.0

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.CATPart`) and Assembly (`*.CATProduct`)

Versions: V5R2 — V5–6R2024 (Windows & Linux)

Table 8: Import Preference Support for CATIA V5 Geometry Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	Yes
Attribute Processing and prefix	Yes - Color, Layer, and Publication
Named Selection Processing and prefix	Yes - Color, Layer, and Publication
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies in the xy plane may be imported
Associativity	Yes
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	Yes
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes
Decompose Disjoint Geometry	No

Notations

This is a Pseudo-Reader (p. 133) in Ansys Workbench. There will be no Ansys pull-down menu in the CATIA V5 program. It is not possible to update from the Active CAD file in CATIA V5.

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The application automatically locks the length unit in the model to millimeters. No adjustment of length unit is necessary or possible. You can change the unit system for display of the Ansys Mechanical application data.

Hidden parts

Parts that are hidden or suppressed in CATIA V5 are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Import CAD Publication as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Publication" or any number of prefixes with each prefix delimited by a semicolon (for example: "NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Publication" or any number of prefixes with each prefix delimited by a semicolon (for example: "NS_ForceFaces;NS_FixedSupports;NS_BoltLoaded").

Using Parameters with CATIA V5 (CADNexus CAPRI CAE Gateway)

- Parameter names contain the entire CATIA Assembly Name and Part Name.
- Parameters are shown at the assembly level, under the Details of Geometry branch in the Ansys Mechanical application.

- Embedded spaces should not be used in the Assembly Name or the Part Name to ensure compatibility with the DesignModeler Parameter Manager.

Caveats and Known Issues

The CATIA V5 associative geometry interface has the following limitations:

Attach to Active CAD Geometry Option

The Attach to Active CAD Geometry is unavailable from CATIA.

Material Properties

Must be assigned to the PartBody or GeometricSet to be imported. If you right mouse and move the Material Property under the PartBody or GeometricSet then they will be imported.

Publications

Publications created by feature selection in the CATIA tree are not supported. Publications created on graphically selected geometry are supported.

WorkPoints

Only standard coordinate-defined points are supported (on Surface, on Line, etc. are not supported).

Small Scale

Small Scale models require the CATIA V5 Geometry Scale to be set to Small Scale. To change the scale setting to allow the import of Small Scale models:

1. Open a native CATIA V5 session
2. **Select Tools**: Options
3. **Select General**: Parameters and Measure
4. Select Scale tab
5. Change Geometry Scale from "Standard Scale" to "Small Scale"
6. Exit native CATIA V5 session
7. **Set environment variable**: `CATUserSettingPath=C:\Users\MyAccount\AppData\Roaming\DassaultSystemes\CATSettings`
8. Now start Ansys Workbench and the CATIA V5 model will import correctly. *Note*: replace "C:\Users\MyAccount\AppData\Roaming" with your %appdata% value

Parameter Units.

Parameter Units (Millimeter, Inch...) are controlled by the CATIA V5 CATSettings.

How to change the CATIA V5 Parameter Units settings:

1. Open a native CATIA V5 session
2. Select Tools: Options
3. Select General: Parameters and Measure
4. Select Units tab
5. Select and change the Units for each of the parameters you want changed (example: Length)
6. Exit native CATIA V5 session
7. Now start Ansys Workbench and the CATIA V5 model will import parameter values using the specified CATIA V5 Units.

Note:

The CATIA V5 Units are saved in the CATSettings which are located at **CSIDL_APP-DATA\DassaultSystemes\CATSettings** and referenced by CAPRI using the variable **CATUserSettingPath** which is setup in the CAPRI environment file (example: CAPRI_CATIA_P3.V5R22.B22.txt). To use a "non-default" CATSettings directory, change the CATUserSettingPath setting in the CAPRI environment file.

Troubleshooting

See the CATIA Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

CATIA V6 Reader (*.3dxml)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the CATIA V6 system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.3dxml`) and Assembly (`*.3dxml`)

Versions: R2010x — R2024x (Windows)

Table 9: Import Preference Support for CATIA V6 Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color and Layer
Named Selection Processing and prefix	Yes - Color and Layer
Material Processing	No
Analysis Type	3D - Yes 2D - Yes- surface and line bodies in the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	No

Notations

Selective Update

This interface does *not* support the Selective Update feature.

Length Unit

The application automatically locks the length unit in the model to millimeters. No adjustment of length unit is necessary or possible. You can change the unit system for display of the Ansys Mechanical application data.

Hidden parts

Parts that are hidden or suppressed are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").

Caveats and Known Issues

The CATIA V6 Reader has the following limitations:

Only supports CATIA V6 - 3DXML with Authoring

There are two types of 3DXML files which can be generated and exchanged. The “3DXML with authoring” (which include the export of geometry and meta-data) is supported by the CATIA V6 Reader. The “3DXML for review” is not supported by the CATIA V6 Reader.

Body Naming

Body hierarchy information is not available in all models. In some models. a generic label will appear in the tree.

Troubleshooting

See the CATIA Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

Creo Elements/Direct Modeling (*.pkg, *.bdl, *.ses, *.sda, *.sdp, *.sdac, *.sdpc)

The interface works in both a Plug-in (p. 133) and a Pseudo-Reader (p. 133) mode.

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.pkg`, `*.bdl`, `*.ses`, `*.sda`, `*.sdp`, `*.sdac`, `*.sdpc`,

Table 10: Import Preference Support for Creo Elements/Direct Modeling Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	No
Named Selection Processing and prefix	Yes
Material Processing	No
Analysis Type	3D - Yes 2D - Yes- Only surface bodies in the xy plane will be imported
Associativity	Yes
Coordinate Systems	Yes
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes - for parts that include both Solid and surface bodies AND the Import Solid and Import Surface preferences are set to Yes
Decompose Disjoint Geometry	No

The Creo Elements/Direct geometry interface also supports the Selective Update feature.

Parts that are hidden or suppressed in Creo Elements/Direct Modeling are not skipped automatically, because of a limitation in the CAD's API.

Due to a limitation in the Creo Elements/Direct Modeling API, mixed import resolution will fail to correctly filter multi-body parts when one body type is owned by another body type.

SES files are not portable between different versions of Creo Elements/Direct Modeling. They should be limited to use on a single machine.

Named Selection Processing

This geometry interface will group each body into Named Selections based upon color when the keyword "Color" is entered as a Named Selection Key. The resultant selection groups will be named with the form, Color:R.G.B. Here R, G and B are values ranging from 0 to 255, which represent the amount of color saturation of Red, Green and Blue color components. For example Color:0.0.0 is black, Color:255.255.255 is white, and Color:255.0.0 is red.

Color Processing

Within Mechanical body colors can be visually altered to match what is displayed in the Creo/Elements Direct session by executing the following steps:

1. Prior to attach/refresh, enable the Attributes preference.
2. Add the keyword "Color" to the Attributes Key list.
3. After attach/refresh is complete, within Mechanical perform Tools->Run Macro and designate this script to be run: `<install dir>\ [version]\aisol\DesignSpace\DSPages\macros\BodyColorByCADColorAttribute.js`

Notations

The Plug-in (p. 133) will import all parts in the model based on body type import filters. Active CAD session models imported from Creo Elements/Direct Modeling can only be updated from an active session unless the model is re-linked to a specified file. A model imported based on its file can only be updated from the file unless re-linked to an active session. Attempting to import a Creo Elements/Direct Modeling file from disk when a session of the CAD is opened will fail so as to not corrupt the active model.

The absence of an Ansys section in the Add-In Modules fly-out menu indicates that the Creo Elements/Direct Modeling Plug-in (p. 133) is not loaded. To load the Plug-in (p. 133) into Creo Elements/Direct, you must first open Creo Elements/Direct Modeling and select Modules under the File Ribbon Tab. Select the Add-Ins item and check the box for Ansys. This will load the plug-in for the current session. To have the plug-in loaded on start-up of subsequent sessions select the Startup button, highlight Ansys and select Add. The Add-In manager will then list Ansys 2020 R1 in the Startup Sequence. Note that because the Ansys menu does not appear after installing the plug-in, you must use the preceding steps for loading on start-up. If the plug-in is not available in the Creo Add-In manager you will need to run the Product & CAD Configuration Manager (p. 24).

If a ".NET initialization failed" error message is seen while activating the Ansys 2025 R1 module, follow the steps below to fix it:

1. From a command prompt navigate to the Creo Elements/Direct installation directory. There will be a bin <platform> underneath that, cd into it. Then type the following command: `SolidDesigner.exe -register`
2. Rerun Product & CAD Configuration Manager (p. 24) for CED.

Configuration Considerations

Due system registry restrictions with Creo Elements/Direct Modeling, the Product & CAD Configuration Manager (p. 24)'s option to configure for "current user" will impact the Geometry Interface's availability for all other users if the current user is the admin.

Limitations

The following limitation is specific to the Ansys 2025 R1 and 2024 R2 Associative Geometry Interfaces to Creo Elements/Direct. There is a known library conflict with earlier releases of this Plug-In, which causes the 2025 R1 Ansys Plug-In to fail loading into Creo Elements/Direct when a previous release of the Plug-In is configured to load into Creo Elements/Direct.

Use the following procedure to enable these versions to be loaded side-by-side in Creo Elements/Direct.

1. Open Creo Elements/Direct and choose File >> Modules
2. Edit the Module Startup List
   - Remove All Ansys Addin's from list
   - Add 2025 R1 Add-in
   - Add All other Ansy Add-ins.

Creo Parametric (*.prt, *.asm)

For more information, see:

Creo Parametric Reader (*.prt, *.prt.*, *.asm, *.asm.*)

Creo Parametric Associative Geometry Interface (*.prt, *.asm)

Creo Parametric Reader (*.prt, *.prt.*, *.asm, *.asm.*)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the Creo Parametric (formerly Pro/ENGINEER) system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the and Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.prt`, `*.prt.*`) and Assembly (`*.asm`, `*.asm.*`)

Versions: Pro/Engineer 19.0 — Creo 11.0

Table 11: Import Preference Support for CREO Parametric Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color and Layer
Named Selection Processing and prefix	Yes - Color and Layer
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	Yes
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only.

If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden Parts

Parts that are hidden or suppressed in Creo Parametric are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Caveats and Known Issues

The Creo Parametric Reader has the following limitations:

Limited Support for Layer Information

Only layers in Creo Parametric faces are read. Creo Parametric bodies and edges do not have layer information. Layer information is not supported for Pro/E Wildfire 4.

Limited Support for 2D Sketch, Cosmetic Groove and Curve Features

The Creo features 2D sketch, cosmetic groove, and curve using equation and local push are not supported.

Limited Support for Suppressed Entities Option

The Suppressed Entities option is not supported at Part level.

Limited Support for Multiple Lumps or Lump with Multiple Shells (lump with void).

The Pro/E Reader does not support multiple lumps in a single body or a lump with multiple shells (lump with void) in certain scenarios. The Reader might not create expected results in some complex BREP scenarios.

Assembly Level BREP or Free Parts

Creo Assembly files may include BREP (such as quilts, free curves, and free points) along with instances. These are called "Free Parts". The Pro/E Reader component does not support Free Parts.

Reading Manufacturing Information from Assembly Files

Pro/E Reader does not support reading PMI data and Hole feature from assembly files.

Limited Support for Assembly Cut Feature

Pro/E Reader supports the assembly cut feature for subtraction operations only.

Limited Support for Simplified Representation

The Pro/E reader does not support part level Simplified Representation. The default SimpRep cannot be used when Derived Exclude rep is used at the part level in a user-defined rep at the sub-assembly level.

Limited Support for Assembly Color and Transparency Property

The Pro/E Reader does not support root assembly level Color and Transparency property.

No Support for Flexible Assemblies

Pro/E Reader does not support flexible assemblies. It will not read instance specific transformation or other information from flexible components. Thus, positioning and/or sizing of components might be incorrect in case of flexible assemblies.

Limited Support for Working Coordinate Systems

Pro/E Reader does not support Assembly level working coordinate systems; only Part level working coordinate systems are supported.

Captures Support

A Creo File May Contain More Than One Capture Set (Combination View Set). You can add Capture-Set-specific annotations. However, when saving a Creo file, only one Capture will be in Active State. In that case, Pro/E Reader will translate only Active State Annotations. If Active state Annotations are added in different captures, then only those Graphical Annotations can be seen in those related Captures.

LCS from Only “Part” Files Are Translated

LCS from only "Part" files (.prt files) is translated and written to corresponding .sat files while reading assembly files by XML route. However, LCS of "Part" files is not translated while reading assembly files by flatten route.

User Defined Attributes

Pro/E Reader will not support user-defined attributes created under Feature, Inherited, Annotation element, Quilt, Surface, Edge, Curve, Composite curve, and Material level. Only those user-defined attributes (call "Parameters" in Creo) defined at the part level will be translated. Pro/E Reader will not support user-defined attributes with Unicode characters.

Limited Support for Material Properties

The custom system of Unit is not supported. The local modification (independent of file level) in a unit of material property is not supported.

Limited Support for Hidden Entity

The Creo modeler allows for hiding merge features, but the Creo viewer does not update that information. Hence, Pro/E Reader does not guarantee correct translation for this hidden feature. Pro/E Reader will not support a Hidden flag on a Solid added to a Layer by a SolidGeom filter in a Creo part file.

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

See the Creo Parametric Errors Related to CAD Integration (p. 129) section in CAD Integration Troubleshooting (p. 127) for detailed information.

Creo Parametric Associative Geometry Interface (*.prt, *.asm)

Before installing the Ansys application, install Creo Parametric and run the program at least once under the user login that you will use to install the Ansys Workbench.

The interface works in both a Plug-in (p. 133) and a Pseudo-Reader (p. 133) on Windows.

The existence of the Plug-in (p. 133) is recognized by Creo Parametric based upon an entry in the configuration file (config.pro) referencing a Pro/Toolkit registry file located in the Ansys Workbench installation (WBPlugInPE.dat). This is configured during the installation process whenever you select Creo Parametric as a geometry source, but can be modified to enable/disable the Plug-in (p. 133) by using the Product & CAD Configuration Manager (p. 24). The Product & CAD Configuration Manager (p. 24) can also be used to change the Creo Parametric installation utilized by the Plug-In. If Creo Parametric was not selected in the original installation then the Associative Plug-In will not be available for configuration. In this case you can run the Product & CAD Configuration Manager (p. 24) and select Creo Parametric under Ansys Geometry Interfaces to make the Associative Plug-In available. Each version of Creo must be configured for the correct version of the geometry interface. Therefore, when switching your associative geometry interface configuration from one version of Creo to another you must unconfigure the old version before configuring the new version. For example if you were configured to work with Creo 7.0.0.0 and decide to migrate to 8.0.0.0, you must use the Product & CAD Configuration Manager (p. 24) to unconfigure 7.0.0.0 before configuring 8.0.0.0.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Product and Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.prt`) and Assembly (`*.asm`)

Table 12: Import Preference Support for Creo Parametric Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes - Datum Curves
Parameter Processing and prefix/suffix key	Yes
Attribute Processing and prefix	Yes - including Color
Named Selection Processing and prefix	Yes - including Color Yes - from Ansys Named Selection Manager (p. 25) and programmatically created attributes
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies in the xy plane may be imported
Associativity	Yes
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	No
Instancing	Yes
Smart Update	Yes
Enclosure and Symmetry Processing	No
Mixed import Resolution	Yes - for parts that include both Solid and surface bodies AND the Import Solid and Import Surface preferences are set to Yes
Decompose Disjoint Geometry	Yes (for use with faces and bodies)
Import Facet Quality	Yes

Notations

The Creo Parametric geometry interface also supports the Selective Update feature.

Assembly parts and sub-assemblies that are hidden or suppressed in Creo Parametric are automatically excluded from import into Ansys Mechanical and DesignModeler.

If you switch between the primary and simplified representations of your geometry in Creo Parametric, you must use the `Erase Not Displayed` option in Creo Parametric to remove the geometry not shown in the Creo Parametric window (but resident in memory) before attaching or refreshing in any Ansys consuming application, such as Mechanical, SpaceClaim, or DesignModeler, Discovery, or Fluent.

Importing Feature Types as Line Bodies

The Datum Curve feature types can be imported from Creo Parametric into Ansys Workbench's Mechanical, SpaceClaim Direct Modeler, or DesignModeler, Discovery or Fluent application as line bodies.

Length Unit

The length unit specified in the Creo Parametric part or assembly is transferred into Ansys Workbench. It is preferable that assemblies and their component parts be constructed using the same length unit. However, when the Geometry Interface encounters a sub-assembly or component part with units different than the top level assembly it will attempt to scale that part to match the units of the root assembly. All converted assembly components will be noted via a warning in the Workbench messages panel. In the event that this unit conversion fails, the following steps can be taken in Creo Parametric to achieve consistent units among all assembly components:

1. Open the assembly within Creo Parametric
2. Open the Model Properties Utility (File> Prepare> Model Properties)
3. Specify the desired units by choosing Change Units, then select the desired unit type and check “Include Submodels”
4. Upon finalizing unit choice via the “Set” button, accept the required field informing you that the “Convert Dimensions” option is being used.

When using an assembly's offset as a parameter, a negative value will cause the direction of the offset to flip and the value will be returned with a positive sign.

Caveats and Known Issues

In no particular order, the caveats and known issues are:

Security Warning

When starting Creo 7 or later an alert may be issued about ac4pro or WB<version>PluginProWF being from an unknown source. These can be addressed by selecting Yes and checking "Remember this decision".

Coordinate System Naming

When importing a Creo Parametric assembly with a coordinate system, default names starting with numbers are replaced with new names. During the import, typically a warning message is displayed that reads, "feature name not valid due to invalid characters."

In addition, the list of coordinate systems is not updated after refreshing the import operation.

Capturing a Body in a Named Selection

Solid bodies can be selected through the Named Selection Manager with Creo versions 7.0 and newer.

However, for previous releases of Creo the Named Selection Manager (p. 25) does not directly offer a mechanism for adding bodies to a Named Selection Group. However, any solid bodies, features, datum curve or quilts contained within a Creo Parametric Layer or Feature Group matching the named selection key will have its faces, edges, and or lines included in a Named Selection or Selection Set with the same name as the Creo Layer or Group. When importing to Mechanical, each Named Selection containing solid or surface bodies will be divided into two separate Named Selections. One containing each body's faces, the other containing edges. When the Named Selection Key is left blank, layers and groups will not be processed for named selection. The method of defining named selections requires explicitly matching the prefix of the layer or group name. If you wish to capture all Creo layers as named selections use the Named Selection Key "Layer". If you wish to capture all Creo Feature Groups as named selections use the Named Selection Key "Group".

Surface Body Naming Convention

Surface bodies imported into the Ansys DesignModeler, Ansys SpaceClaim Direct Modeler, and the Ansys Mechanical application include numerical references to the parent part or assembly and Creo Parametric quilt ID. For example, a part named H103 with three Creo Parametric quilts 1, 2, and 3 will be identified as H103[1], H103[2] and H103[3].

Layer Considerations

Solid or Quilt bodies belonging to a hidden Creo layer will be not be imported when the layer is hidden.

Note that changing layer visibility in Creo Parametric does not mark a part/assembly as modified. To pick up such a change in state when importing to Mechanical, SpaceClaim or DesignModeler the Geometry cell's Smart Update preference must be disabled.

Updating Instances

Model Instances

If you wish to update a particular instance of a model, it is necessary that the active model in Creo Parametric be the desired instance, otherwise a generic version of the instance will be used for the update.

Part Instances

Within the Creo Parametric assembly, any component impacted by any assembly level feature will not be processed as an instance (p. 133), but the full geometry will be imported. Note that for models where an assembly feature was modified so that component(s) are no longer impacted, you may still be required to update the intersection information manually otherwise the Plug-in (p. 133) will not be able to process them as instances. The following procedure describes the process of manual update of intersection data for Creo Parametric:

1. Enable feature visibility (Model Tree> Settings> Tree Filters> Display Features)
2. Select Assembly Feature for Edit
3. Navigate to the feature's Intersection tab
4. Disable Automatic Update
5. Remove all components in the list
6. Check Automatic Updates
7. Accept the feature changes

Material Data

Material data can be transferred into Ansys Workbench provided one or more of the material properties are defined for the Creo Parametric material. The properties for a Material are Density, Poisson's Ratio, Young's Modulus, Coefficient of Thermal Expansion, Specific Heat Capacity and Thermal Conductivity.

File Versions

Multiple versions of Creo Parametric files are designated with a `.#` extension that is appended to the `.prt` or `.asm` extension (for example, `wrench.prt.1`, `wrench.prt.2` are names of two Creo Parametric file versions). You can access a specific version of a Creo Parametric file through the Ansys Workbench Project Schematic.

You are strongly advised to avoid stripping the version numbers from Creo Parametric files for the following reasons:

- If you open a stripped version of the file in Creo Parametric and import the file via the Plug-in (p. 133), it will indicate the wrong version if there is any other version of that part file in the same directory.
- If you attempt to load a stripped version of the file by using the Pseudo-Reader (p. 133), the highest file version of that model will be loaded.

Regarding the updating of Creo Parametric file versions, updating will be version specific if you use the reader or if you do not open a copy of that model with the same name in a Creo Parametric interactive session. However, if you open one version of a model in Creo Parametric and you request

the same model, but a different version for attach or update, the currently active version will be assumed to be the one that you want. This assumption is necessary for the following reasons:

- Creo Parametric does not allow two different versions of the same part to be active at one time, using the same name.
- If you save a model in a Creo Parametric session, its version is incremented (for example, if you attach via the Plug-in (p. 133) with version 3, then save the model in Creo Parametric, the version of the active model would be version 4).

Configuration Considerations

If an active Creo Parametric file does not appear under Active CAD Files, either Creo Parametric is not running or the Ansys Workbench Plug-in (p. 133) for Creo Parametric is not loaded. For this last condition, run the Product & CAD Configuration Manager (p. 24). See Using the CAD Configuration Manager in the Ansys, Inc. Windows Installation Guide for detailed information.

On some Windows systems, where the Plug-in (p. 133) was configured for only the current Administrator, the Ansys Geometry Interfaces to Creo Parametric are not loaded as expected after being selected for install. This is because Creo Parametric is not detecting the required configuration file in the user's login folder (either %HOME% or %HOMEDRIVE%%HOMEPATH%). **Work-around:** Copy the file named config.pro from your login directory to Creo Parametric start-up directory which can be determined by either examining the properties for the shortcut used to launch Creo or by running `<path to Creo Parametric install>\bin\parametric.bat`, or by performing the following within Creo: `File >> Manage Session >> Select Working Directory within Creo`.

On the Windows platform, users without write access to their Ansys Workbench installation may encounter the following error when attempting to import Creo Parametric models without an active CAD session: "No write access, please choose another start-up directory for trail file creation". This issue can be addressed by specifying an alternate location for Creo Parametric to create trail files. The *trail_dir* directive for the config.pro file is the mechanism offered by PTC. For example, adding the line *trail_dir $TEMP* to the config.pro file will force Creo Parametric to create trail files in the location referenced by the TEMP environment variable.

Administrative users should avoid configuring the Creo Parametric Associative Geometry interface both for “All Users” and “Current User”. Non-Administrative users should avoid configuring the interface for the same Creo installation that an Administrator has already setup for “All Users”. When a non-Administrative user configures for a Creo version different than that already configured for all users, that user’s import will work normally for only the CAD version specified by the non-admin user.

Creo Parametric can potentially leave behind the Ansys ribbon tab after unconfiguring the Plug-In. In this case, the ribbon tab can be manually removed by deleting `%APPDATA%\PTC\ProENGINEER\Wildfire\.wf.\Settings\creo_parametric_customization.ui`. In cases where the config.pro directive “load_ui_customization_run_dir” is set to yes this file will instead be located in the Creo start in directory.

This geometry interface is supported with a locally setup version of Creo Parametric using the CAD's official installer. It executes with a properly configured network installation of Creo Parametric, for specifics on complying with network requirements see the Creo Installation & Administration Guide: *Strategies for the Mass Deployment of Creo Applications*.

Decompose Disjoint Geometry

Creo Parametric supports surfaces with multiple external contours. Prior to Workbench 14.0 these were represented as a single, disjoint face when transferred to the Ansys Mechanical application. To add greater flexibility in the Ansys Mechanical, DesignModeler SpaceClaim Direct Modeler, Fluent, or Discovery application and preserve unique IDs, a preference has been added to break apart the Creo Parametric surfaces into a single face for each external contour during import. In DesignModeler SCDM, and Discovery decomposition will always be attempted. In Mechanical, decomposition is the default pre-selected behavior, but can be disabled when desired.

Associativity will be maintained for sub-faces as long as the edges composing the external contour are static. Adjusting the sketch elements from which the external contour is defined, such that their internal identifiers change, will likely result in lost associativity.

There is a significant slowdown of Decompose Disjoint Geometry feature in Creo 7.0 when operating in multi solid body mode. Efforts are underway with PTC to substantially speed-up this functionality. The two work-arounds are:

1. Use Creo's Split feature to separate each disjoint body into a body with one collection of connected faces, and turn off the decompose preference.
2. Disable multi-solid mode in Creo by adjusting the config.pro settings:
   - enable_multibody no
   - smt_creo7_multibody no

Decompose Disjoint Solids Instability

A very limited set of models have been observed to cause Creo Parametric to stop running when this preference active. The resolution is to upgrade to Creo 7.0.1.0 or later, where PTC has resolved a problem in their sorting algorithm.

DesignModeler Considerations

When a Creo Parametric model does not attach in the DesignModeler application but does attach in the Ansys Mechanical application,

1. A possible cause is a Round or Fillet radius in Creo Parametric that is failing to translate. You may find the cause by suppressing some or all of the Rounds or Fillets in the Creo Parametric model and then try to attach in the DesignModeler application.
2. Creo Parametric Model tolerance (Accuracy) is too loose. You may be able attach the model successfully by setting Creo Parametric to use Absolute Accuracy along with a tighter tolerance.

To do so, edit the file named "config.pro" in the Creo Parametric.

1. In Config.pro, add:

   ```
   enable_absolute_accuracy yes

   accuracy_lower_bound 1.0e-7
   ```

2. Open the model in Creo Parametric

3. Accuracy>Absolute>[Enter new value]>-->-->Regen>Yes

   For example, if the Absolute Accuracy Value is [1.2000e-03 mm], then try entering a new value that is 0.5 times that [0.6000e-04 mm]

Future Creo Parametric models can be created with tighter tolerance from the start (two orders of magnitude tighter than default) although may result in increased memory use and diminished performance. Note that after tightening a model's tolerance it can fail to regenerate. If so, you can attempt less restrictive values until you find the one that will regenerate and translate into the DesignModeler application. However, not every model can regenerate at a tight enough tolerance to successfully translate.

In cases where adjusting the absolute tolerance does not work, you may need to defeature parts of the model until it imports successfully.

Parametric Updates

There is a known problem performing parametric updates of Creo Parametric models which contain parameters with "locked" access. The Creo Parametric API does not provide information regarding the access state of parameters and the Creo/Toolkit does not detect assembly parameter relationships driven by dimensions. This can result in refresh failure. **Work-around:** Ensure that locked parameters do not have a prefix/suffix corresponding to the existing parameter filter to prevent their import.

When working in Creo Parametric's Simulation environment (Applications->Simulate) specifying an out of range parameter value can leave the CAD model in an undesirable state following update.

Smart Update

In order for the Geometry Interface to effectively identify and skip retrieval of unchanged assembly components the following requirements must be met:

1. Mass Density must be defined for all assembly components.
2. Model must be fully regenerated.
3. Model must be saved before being retrieved using Reader mode (Creo Parameteric session is not active at time attach/refresh is initiated).

Note:

Mass density can be applied to assembly components without a preexisting setting by either of the following methods:

- **Either**: File> Prepare> Model Properties> Mass Properties> Change > Specify Density Value -> Generate Report. Then accept the prompts for each component to which the value is to be applied.
- **Or**: Defining a model parameter MP_DENSITY and specifying the value in the column next to each component. The model parameter can be created from Setting Menu> Tree Columns> Type: Model Params> Name: MP_DENSITY> Click right arrow to move into Displayed Column.

2D Analysis Considerations.

Some surfaces which appear eligible for 2D analysis in Creo Parametric may be filtered during import, because their Z extent is beyond tolerance from the XY plane. Tolerance is considered to be 10e-5 multiplied by the surfaces diagonal length.

Using Seed and Boundary with Named Selections

The Associative Geometry Interface supports the Creo's Seed and Boundary selection technique as a means of adding groups of surfaces to a Named Selection. Seed and Boundary selection can be performed as follows:

1. Choose "Geometry" as the Creo selection filter.
2. Select a surface as **Seed**
3. Press **SHIFT**-Key (keep pressed)
4. Select another surface, or set of surfaces, as **Boundary**
5. Release the **SHIFT**-Key

The result is that all surfaces from **Seed** to the **Boundary** are selected excluding the Boundary surface. These can then be added to a Named Selection via the Create and replace options in the Named Selection Manager.

Limitations

Creo Platform Agent Corrupt

When the installation of this PTC utility is broken it will block the startup of the Ansys Creo plug-in. To work around this problem you can either (a) set the environment variable PROTK_DELAY-INIT_NO_DELAY=TRUE, or (b) reinstall Creo to correct the problem with Creo Agent.

Creo Weld Import

Starting with Ansys 2019 R1 it is possible to import solid bodies belonging to welds, with the following limitations:

- The Named Selection Manager will appear to insert solid weld geometry into a Selection Set, but this geometry will not be included in the set when translated to SpaceClaim Direct Modeler (SCDM), Mechanical, or DesignModeler. The same limitation applies to geometry contained in Surface welds.
- Solid, Surface and Light weld geometry are included in a Named Selection when added to a Layer or Feature Group.
- Prior to 2022 R1, light or surface weld body was named to match the part or assembly which contained it. Beginning in 2022 R1, these bodies are named to match their Creo Weld Feature.
- Starting with 2022 R1, all Light Weld curves will be combined into a single line body, previously the translator created one body per curve.

Library Conflict

The following limitation is specific to the Ansys 2025 R1 and 2024 R2 Associative Geometry Interfaces to Creo Parametric. There is a known library conflict with earlier releases of this Plug-In, which causes the 2025 R1 Ansys Plug-In to fail to load into Creo Parametric when a previous release of the Plug-In is already loaded into Creo Parametric.

Use the following procedure to enable theese versions to be loaded side-by-side into Creo Parametric via Manual Method.

1. Open the config.pro file containing the registration of your Creo Parametric Associative Geometry Interface. This will be located one of the following locations:

   - When configured for just this user: %HOMEDRIVE%%HOMEPATH%

   - When configured for all users of this Creo Parametric Installation: <Path to Creo>\Common Files\text\

2. Reorder any PROTKDAT lines referring to the Ansys file WBPlugInPE.dat, so the line referencing v251 is listed in the file before all other Ansys lines in that file.

3. Save and close the file.

Use the following procedure to enable theese versions to be loaded side-by-side into Creo Parametric via CAD Configuration Method.

1. Open Ansys CAD Configuration Manager (CCM) for each version of the Plug-In desired to run.

2. Unconfigure the Creo Parametric Associative Geometry Interface

3. Use CCM 2025 R1 to configure the Geometry Interface.

4. Use CCM to configure the Geometry Interface for all other versions of Ansys.

Mixed Unit Support

It is recommended, but not required that all parts and sub-assemblies should be in the same unit as the root assembly. Models with mixed units imported into the Ansys Mechanical application will trigger a warning indicating the mixed unit. Note that Siemens NX and PTC Creo are the only CAD systems that can provide mixed unit assembly data correctly to Ansys Workbench.

Relationship Driven Sketch Parameters

Feature parameters linked to relation defined sketch or section dimensions will import to Workbench as Independent parameters due to a known problem in Creo. Any attempts to refresh/update the model with a changed value for such parameters will result in either an unexpected parameter value or aborted update due to a Creo regeneration failure.

Smart Update of Mixed Unit Assemblies

There are some situations where assembly components will fail to be smart updated due to the component(s) having different unit system than the root assembly. The work-around for this issue is to normalize model units with in Creo, see the *Length Unit* section of the Creo Parametric Associative Geometry Interface help for detailed instructions.

Unexpected Entities in Named Selection

In some situations, entities removed from a Creo box or group selection may not be excluded from a Named Selection creation/edit action. In such cases the work-around is to individually select desire entities.

Windchill Conflicts

The environment variable PTC_WF_ROOT, used to define the default workspace cache location, can break import models not residing in the Creo start-in folder. The workaround is to us PTC_WLD_ROOT instead of PTC_WF_ROOT.

DesignModeler (*.agdb)

Ansys DesignModeler topics:

- Face Geometry Transfer (p. 76)
- Specific to Ansys Mechanical (p. 78)

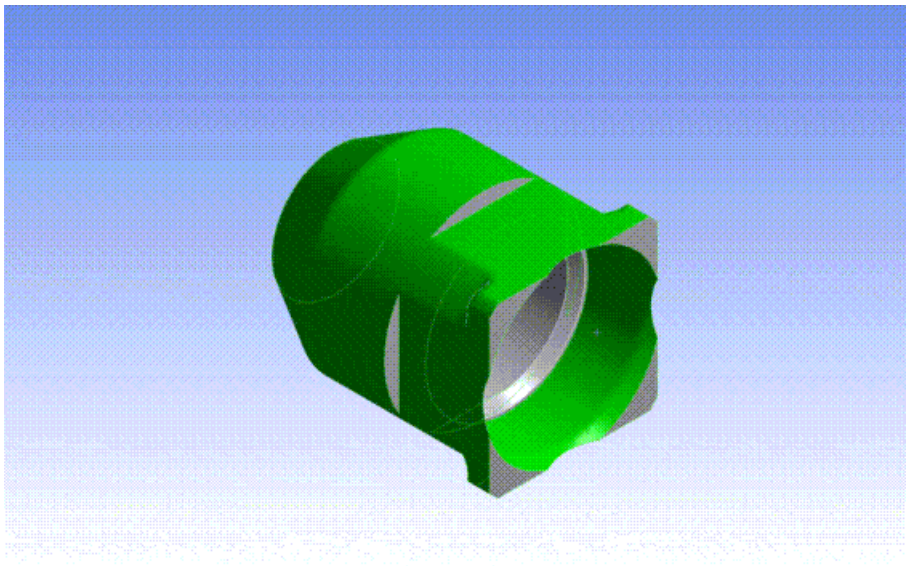
Face Geometry Transfer

Faces without associated geometry may be generated when attaching to a CAD system or when importing from a file. This may occur for all CAD systems or file formats with the exception of Parasolid (*.x_t, *.xmt_txt, *.x_b, *.xmt_bin) (p. 95), IGES (*.igs, *.iges) (p. 82), STEP (*.stp, *.step) (p. 116), Monte Carlo N-Particle (*.mcnp) (p. 86), Ansys BladeGen (.bgd) and Ansys DesignModeler (.agdb). Loading geometry into the Ansys DesignModeler application may result in a greater frequency of occurrence of such faces than when loading geometry into the Ansys Mechanical application.

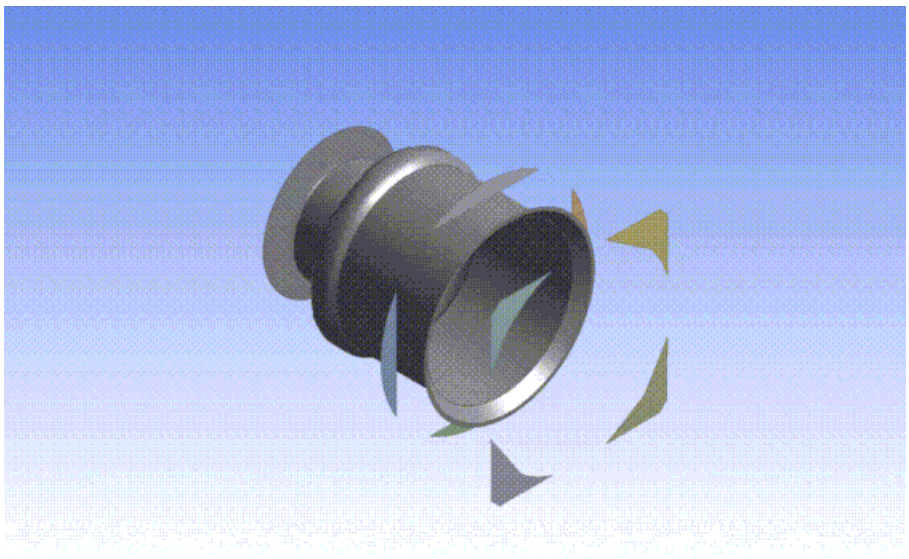
The Ansys Mechanical application will fail to construct a part when an associated face without geometry is found and will report an error.

The Ansys DesignModeler application will construct parts excluding faces without geometry. A solid or surface body in the CAD system or file with such faces detected will result in surface bodies being constructed. The constructed surface bodies are generated from the faces with geometry and will consist of the faces that are connected by shared edges. If the Line Bodies property is set to "Yes" (enabling import of line bodies), then line bodies will be constructed from edges only belonging to faces without geometry that have been removed. A warning message indicating a partial import will be generated for all parts that have been imported in Ansys DesignModeler with faces without geometry having been removed.

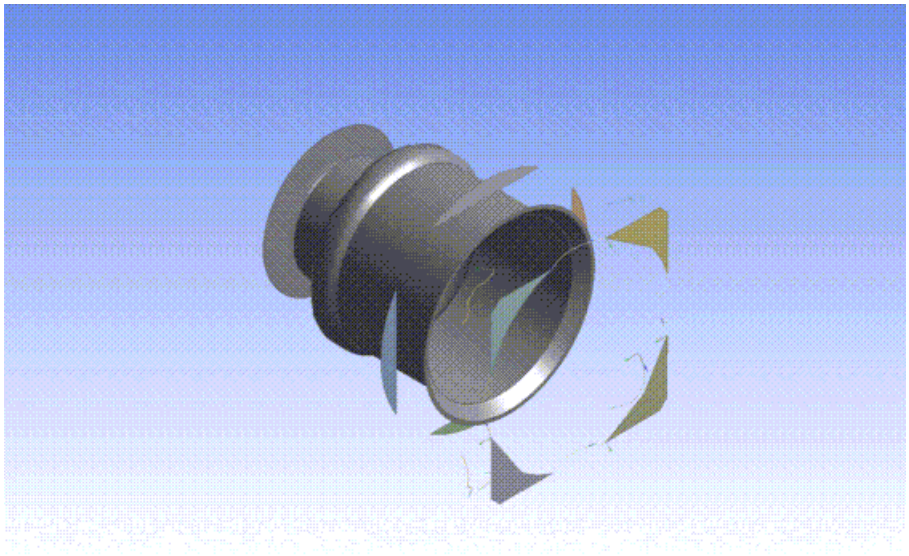
Example 1: Face Geometry Transfer Scenarios



Shown above is a body with two faces highlighted for which geometry is not constructed during transfer.



Shown above is a body in the Ansys DesignModeler DesignModelerapplication where two faces are not being transferred.



Shown above is a body in the Ansys DesignModeler application where two faces are not being transferred and the Line Bodies property is set to yes.

Specific to Ansys Mechanical

The Ansys Mechanical application supports the DesignModeler application as both a pseudo-reader and a plug-in.

Attaching Geometry

Unlike other CAD systems, filtering of bodies during import from DesignModeler to the Ansys Mechanical application is not done through import options. All unsuppressed bodies in Ansys DesignModeler will be transferred to the Ansys Mechanical application.

Any planes in the Ansys DesignModeler application that are flagged to be exported will appear in the Ansys Mechanical application as a local Cartesian coordinate system upon attachment and will be inserted into the Ansys Mechanical application tree. The automatic creation of the coordinate system occurs during attachment and is updated during a refresh.

Any enclosures defined with a model in the Ansys DesignModeler application will be retained upon attaching. Once in the Ansys Mechanical application, the enclosure has the following characteristics:

- A transparency value of 0.1 (subdued) is assigned to the enclosure to assist in the visualization of the model items inside the enclosure.
- A default field material of Air is automatically assigned to the enclosure. You can change the default field material in Engineering Data. A field material is used in any open domain simulation where an artificial boundary such as an enclosure is used to surround the model. Any fields extending from the model to the edges of the enclosure experience the field material.

You can control whether Coordinate Systems and Enclosures that are created in the DesignModeler application files are imported in the Ansys Mechanical application by checking the Advanced Geometry Options (p. 14), specifically:

- Import Coordinate Systems
- Enclosure and Symmetry Processing

Parts and Assemblies

The Ansys DesignModeler application does not support assemblies, but does allow you to create and import multiple bodies. If you import a model made up of independent multiple bodies, they will be imported into the Ansys Mechanical application as individual bodies using contact elements.

If you import a model that has multibody parts (that is, parts that include multiple bodies within them as a group), then the model will be imported using shared topology so that no contact is required and meshes on the volume interfaces will match.

The Ansys DesignModeler application parameters are all set at the model level. Since Ansys DesignModeler publishes its own parameters to the project schematic, its parameters are not displayed in the Ansys Mechanical application under the Geometry branch.

For line body properties that appear in the Ansys Mechanical application, the values shown are for the raw cross section, that is, the offset type and/or user defined offset from the Ansys DesignModeler application's body properties have no bearing on the calculations shown in the Ansys Mechanical application. They are taken directly from the cross section without applying any offset.

Surface body thicknesses and material properties are transferred from the Ansys DesignModeler application to the Ansys Mechanical application.

Notations

If you are using the Ansys DesignModeler application as a pseudo-reader, the Ansys Mechanical application will maintain associativity with the Ansys DesignModeler application model. If you modify parameters in the Ansys Mechanical application, the Ansys DesignModeler application model will change upon update.

Models that have fatal errors when regenerating in the Ansys DesignModeler application will not be able to be imported via the reader mode into the Ansys Mechanical application. Before importing, first open the model in the Ansys DesignModeler application to resolve the regeneration failures.

When the Ansys Mechanical application is first attached to an Ansys DesignModeler application, the Fluid/Solid property associated with all DesignModeler solid bodies will be transferred to a Material Assignment property for all associated solid bodies in the Mechanical application. However, when refreshing all data in the Mechanical application, following an initial attach to DesignModeler, the Mechanical application Material Assignment property for solid bodies will not be updated further.

When an Ansys DesignModeler model has multiple material definitions with the same name, only one of these will be transferred to the Mechanical application.

Length Unit

The Ansys Mechanical application automatically sets the length unit to match the units used in the Ansys Workbench project itself, not the length unit of the model in the Ansys DesignModeler.

Specific to SpaceClaim

SpaceClaim does not support the import of non-manifold geometry. If the DesignModeler geometry contains shared topology, it will be rolled back to the state just before the shared topology was created before transferring it to SpaceClaim.

Discovery (*.dsco)

Importing Discovery Files into SpaceClaim

SpaceClaim can import the Discovery file format (*.dsco); however:

Note:

The Discovery files are imported using a translator. A full data match may not be guaranteed in all cases as some data may not be transferred during import.

If the model contains simulation data from Discovery, the Physics objects and existing simulation data will be removed and will be lost when the model is saved.

Also see SpaceClaim (*.scdoc, *.scdocx) (p. 114).

Fusion (*.f3d, *.f3z)

The interface works in both a Plug-in (p. 133) and a Pseudo-Reader (p. 133) mode.

As shown in Table 13 below, some features are currently unsupported, but over time these features will be implemented.

Prerequisites

Use of Fusion requires the installation of the Autodesk license.

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Table 13: Import Preference Support for Fusion Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	No
Parameter Processing and prefix/suffix key	Yes

Attribute Processing and prefix	Yes
Named Selection Processing and prefix	Yes
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes
Associativity	Yes
Coordinate Systems	No
Work Points	Yes
Reader Save File	No
Instancing	Yes
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes - for parts that include both Solid and surface bodies AND the Import Solid and Import Surface preferences are set to Yes
Decompose Disjoint Geometry	No
Import Facet Quality	Yes

Notations

Occurrences, body folders, and individual bodies that are hidden in Autodesk Fusion are automatically excluded from import.

Caveats and Known Issues

Refreshing Local Documents

If Fusion is restarted after attaching a local document in Plug-in mode (not Pseudo-Reader mode), any attempts to refresh the geometry will produce a "Failed to find requested offline document" error. This happens because the geometry source in Workbench is no longer valid. The issue can be fixed by simply replacing the geometry source, and it can be avoided entirely by using online documents rather than local ones.

Works Points Import Recommendation

Work points (called "construction points" in Fusion) are imported under the same assembly/part that they are stored under in the Fusion Browser, not the part whose geometry they are attached to. Work points stored under an assembly/part which is not imported will be imported under the root assembly instead, but these are not visible in Mechanical. Because of this, the recommendation is to move the construction points to the same component/occurrence that they are attached to.

Limitations

Due to a third party conflict between Ansys and Fusion, the 2025 R1 version of the Ansys plug-in may not load on your machine. This issue affects import from both open sessions of Fusion, and using pseudo-reader mode. We are actively investigating the matter and plan to provide a correction in an

upcoming service pack of 2025 R1. In the meantime, if you experience this issue, please contact customer support to inquire about updates and workarounds.

The following limitation is specific to the Ansys 2025 R1 and 2024 R2 Associative Geometry Interfaces to Fusion. There is a known library conflict with earlier releases of this Plug-In, which will cause Fusion to crash when attempting to load the 2025 R1 Ansys Plug-In when a previous release of the Plug-In is configured to load into Fusion.

Use the following procedure to enable these versions to be loaded side-by-side in Fusion.

1. Unconfigure all versions of Ansys Fusion Add-In using the CAD Configuration Manager
2. One-by-one configure all desired versions of the Ansys Fusion Add-In except 2025 R1.
3. Open Fusion >> Utilities >> Add-Ins. Select the Add-Ins tab, pick each DSPlugInAFCOM and uncheck the box: "Run on Startup". Close Fusion.
4. Use CAD Configuration Manager 2025 R1 to configure Fusion.
5. Start Fusion. Open the Add-In's manager as before, select any version of the plug-in not loaded on Startup and click Run.

Troubleshooting

No known issues have been identified in the CAD Integration Troubleshooting (p. 127) section.

IGES (*.igs, *.iges)

The application supports IGES files as a Reader (p. 133).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Linux: Configuring CAD Products> Using the CAD Configuration Manager

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.igs,*.iges`

Table 14: Import Preference Support for IGES Geometry Interface

Import Solids	Yes
Import Surfaces	Yes

Import Lines	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color and Layer (No for DesignModeler)
Named Selection Processing and prefix	Yes - Color and Layer (No for DesignModeler)
Material Processing	No
Analysis Type	2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	Yes (No for DesignModeler)
Reader Save File	No
Instancing	Yes
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes (No for DesignModeler)
Clean Geometry On Import	Yes (No for DesignModeler)
Stitch Surfaces On Import	Yes (No for DesignModeler)
Import Facet Quality	Yes

Notations

Specific to non-DesignModeler products)

Selective Update (Windows)

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden parts

Parts that are hidden or suppressed in CATIA V5 are skipped automatically by this interface.

Import CAD Color as an Attribute of Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute of Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Specific to DesignModeler product

IGES imports sets of surfaces that enclose a region to create a solid body.

IGES models cannot be imported if they contain bodies that extend beyond 500 meters in any direction from the world origin, even when the Large Model Support option is enabled. For IGES imports to be successful, the model must fit inside the base 1 km^3 size box.

Similar in behavior to IGES files exported from Ansys Mechanical APDL, IGES models that do not have a defined unit system in the file will be interpreted as meter units upon import into Ansys DesignModeler and/or Ansys Workbench; that is, 1.0 = 1 meter.

Caveats and Known Issues (Specific to non-DesignModeler products)

No caveats or known issues.

Caveats and Known Issues (Specific to DesignModeler product)

For IGES assembly models containing any assembly level parts and root level parts of the same name, the root level parts with those duplicate names fail to import, unless they are imported into DesignModeler or SpaceClaim Direct Modeler first.

Caveats and Known Issues

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

JT Reader (*.jt)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the JT system be installed.
- No CAD associativity support.
- No CAD Parameters support.

This interface can be configured during Ansys Installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `Part (*.jt)`

Versions: 8 — 11.5

Table 15: Import Preference Support for JT Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes — Color
Named Selection Processing and prefix	Yes — Color
Material Processing	No
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	No

Notations

Selective Update

This interface does *not* support the Selective Update feature.

Length Unit

The application automatically locks the length unit in the model to millimeters. No adjustment of length unit is necessary or possible. You can change the unit system for display of the Ansys Mechanical application data.

Hidden Parts

Parts that are hidden or suppressed are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Caveats and Known Issues

The JT Reader has the following limitations:

JT B-REP representation support. NURBS Surfaces, trimmed by NURBS curves.

JT Tessellation-only data files are not supported. JT Reader requires that the JT file contains Boundary Representation (B-rep) data.

Body Naming. Body hierarchy information is not available in all models. In some models, a generic label will appear in the tree.

Troubleshooting

See the JT Errors Related to CAD Integration in the CAD Integration Troubleshooting (p. 127) section for detailed information.

Monte Carlo N-Particle (*.mcnp)

The application supports Monte Carlo N-Particle files as Reader (p. 133).

- This is a stand-alone reader that does not require an installation of the Monte Carlo N-Particle CAD program.
- No CAD associativity.
- No CAD Parameters.

- The MCNP format contains solid bodies only. Surfaces and lines will be ignored.

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

This is a stand-alone reader, which does not require an installation of the Monte Carlo N-Particle software or a Monte Carlo N-Particle license.

Document import supported by interface: `*.mcnp`

Notations

Monte Carlo N-Particle database files can only be read into Ansys DesignModeler (p. 76). They cannot be imported directly into the Ansys Mechanical application.

Monte Carlo N-Particle database files must be updated with the Generate option prior to proceeding with an analysis.

When importing an MCNP file, the solid body representing the outside world body is automatically hidden.

Caveats and Known Issues

No caveats or known issues.

NX (*.prt)

For more information, see:

NX Reader (*.prt)

NX Associative Geometry Interface (*.prt)

NX Reader (*.prt)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader that does not require the NX system be installed.
- No CAD associativity support.
- No CAD Parameter support.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.prt`) and Assembly (*.prt)

Versions: UG 11 – NX 2406

Table 16: Import Preference Support for NX Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color and Layer
Named Selection Processing and prefix	Yes - Color and Layer
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	Yes
Work Points	Yes
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden Parts

Parts that are hidden, blanked or suppressed in NX are skipped automatically by this interface.

Import CAD Color as Named Selections

To import an object's Color as Named Selections in Ansys Workbench, specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as Named Selections

To import objects that are assigned a Layer as Named Selections in Ansys Workbench, specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").

Caveats and Known Issues

Assembly Cut.

NX 3 implements a new feature named "assembly cut". The Reader provides limited support for this new feature.

Tool Body Used for “cut”.

Reader does not distinguish between the tool body and target bodies.

Limited Support for Assembly Attributes.

Translation of Hidden Attribute in assembly files is not supported. Translation of layer information in assembly files is not supported. When same sub-assembly definition is instantiated more than once and assigned different colors at each level, the Reader will translate the color combination of the first sub-assembly definition colors found. The Reader does not duplicate the same sub-assembly definitions if the sub-assembly is used multiple times.

Limited Support for Coordinate Systems.

WCS present in the part files will be translated only if they are referred through Reference Sets that to be included in the assembly instances. For example, in the default 'Part' Reference Set does not contains WCS. Hence, WCS will not get translated with 'Part' Reference Sets when included in the assembly files. This matches with the NX Modeler behavior.

Hole Feature.

Hole name will not match the feature name in NX. For example, in NX the feature name in a feature tree may be SIMPLE HOLE (3), however, reader outputs it as SIMPLE HOLE (1). Hole top face information is not supported. Pattern of Hole feature is not supported.

Datum Target.

Orientation_Position_1 and *Orientation_Position_2* values for all Datum Targets are not supported.

> ***Important***: *Non-semantic Datum Target without datum reference is not supported.* Semantic Datum Target Type Arbitrary Shape is not supported.

Name Attribute for Sketches.

As for sketches, the name attached to a sketch is not translated, but the name attached to the curve or point of a sketch will be translated.

Material Properties.

If a material property has multiple values, then only the first value in the table is given as the value for that property. Only isotropic materials from the NX material library are translated. In material properties, only the Unit systems available in an NX Modeler are supported. Custom Unit systems are not supported.

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

No known issues have been identified in the CAD Integration Troubleshooting (p. 127) section.

NX Associative Geometry Interface (*.prt)

The interface works in both a Plug-in (p. 133) and a Reader (p. 133) mode.

The existence of the Plug-in (p. 133) is recognized by NX based on the UGII_CUSTOM_DIRECTORY_FILE environment variable. If the variable points to a text file, NX reads the text file and interprets each line as a directory containing a NX third party Plug-in (p. 133). NX loads the Ansys NX geometry interface from the appropriate directory specified in the file.

After launching NX, if the Ansys (version specific) menu is not displayed, run the Product & CAD Configuration Manager (p. 24) with NX selected for configuration (Start->Workbench->Utilities->Product & CAD Configuration Manager (p. 24) in Windows.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

> **Windows**: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

Both part and assembly document import is supported.

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Table 17: Import Preference Support for UG NX Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes — lines consumed by NX features are not imported
Parameter Processing and prefix/suffix key	Yes
Attribute Processing and prefix	Yes - from entity attributes, entity names, colors, and groups
Named Selection Processing and prefix	Yes - from Ansys Named Selection Manager (p. 25), entity attributes, entity names, colors, and groups
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes; surface and line bodies on the xy plane may be imported
Associativity	Yes
Coordinate Systems	Yes - including visible WCS
Work Points	Yes
Reader Save File	Yes
Instancing	Yes, except lines
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes - for parts that include both solid and surface bodies AND the Import Solid and Import Surface preferences are set to Yes
Decompose Disjoint Geometry	No
Import Facet Quality	Yes

Notations

Tabular Material Data

When importing tabular data for a material property, the first data point is used to complete the material definition.

NX Color Attributes

NX color attributes can be imported using the 'Color' key in the list of attribute or the named selection keys. Once the attribute or named selection import is enabled in the WB2 geometry properties window, append the 'Color' key in the list of keys, separated by semicolons, to import the NX color attributes. Note that, beginning with Release 14.5, the option for 'NX Color Processing' is not supported and is removed from the CAD configuration.

Load Options

Load options for a part can be specified during the import. The Plug-in (p. 133) will first attempt to load the options from the options file specified using the UG_LOAD_OPTIONS environment variable. If it fails, an attempt to load the load_options.def file from the part directory is made. If both the attempts fail, a third attempt is made to load the options from %HOME%\load_options.def, where HOME is an environment variable pointing to the user's home folder. See the NX documentation for more information on the usage of load options.

It is recommended that all models for import have only parts that use the same unit system as the assembly of which they are included. This can be achieved by using the ug_convert_part command to convert all parts in an assembly in the same unit. To do this, launch the NX command prompt from the programs menu. For NX continuous release series the command prompt can be found in Programs > Siemens > Tool – NX Command Prompt. Type 'cd<Directory>'to change the directory containing the assembly. Run the command 'ug_convert_part -mm -d -s -y' or 'ug_convert_part -in -d -s -y', without the quotes to change the unit to millimeter or inch, respectively. See the NX documentation on the ug_convert_part command for more information. Assemblies that are imported using parts with units systems that differ from that of the assembly will have those parts scaled to generate a consistent unit system for all components.

Using Multiple Versions of NX

If you have multiple versions of NX installed, you must make sure that the UGII_BASE_DIR and UGII_ROOT_DIR environment variables point to the correct version of NX when running the Ansys Mechanical application in Reader (p. 133) mode.

The NX reader will use the version of NX set by these variables when processing the attach or updates. Parts files are saved in the format designated by the above two variables.

Maintaining Associativity of Persistent IDs

Object associativity is of particular importance if you update/refresh a geometry with applied loads and supports. The Ansys NX geometry interface uses the User Defined Objects (UDO) to store persistent IDs to maintain the associativity of the geometry between NX and the Ansys Mechanical application. The part file must be saved after import to store this data in the NX model. This can be done by saving the file in NX after an attach via an NX Plug-in (p. 133) session, or with the use of the 'Reader (p. 133) Mode Saves Updated File 'option within the Ansys application when using the Reader mode for import. The 'Reader Mode Saves Updated File' option can be found under the 'Advanced Geometry Options' group on the Ansys Workbench Project page. The part file will be saved at the end of an import process using the same file name in the same directory. The current part file will be backed up by changing the extension of the file to **bak** before saving the part. Make sure that the file is not set to read-only.

Parametric update also requires the UDOs to have been saved in the NX model. It is important to note that if you make parametric changes those too will get saved and the model will be modified from the original version.

If you do not save the parts files, during an update after the initial attach, the loads and supports could disappear or be applied to the wrong entities.

Facet Smoothness Control

The geometry faceting obtained by the Ansys NX geometry interface is view-dependent. That means the facet smoothness of the imported geometry depends on how the object is displayed on the screen. Although the default settings are generally acceptable for most of the cases, these

may not be adequate for a geometry needing smoother facets. The facet smoothness in such cases can be controlled by the ANS_UG_FACET_AFFINITY environment variable.

The variable can be set to any floating point number from 1e-8 to 1e+8. The higher the value is, the finer the facet will be and the longer it will take to import the geometry. If the variable is absent or outside the acceptable range, 1.0 will be assumed as the default value for it.

Note that this environment variable may get removed entirely or replaced by a more user friendly variation in the future release.

Caveats and Known Issues

Imported Body Size

All NX bodies must be within a 1000 x 1000 x 1000 meter cube, centered about the origin of the absolute coordinate system.

Initial Launch

After installation of Ansys Workbench, importing an NX model using the Reader (p. 133) mode requires the CAD be launched one time. Otherwise, the first attempt to import will fail, but subsequent attempts will work normally.

Large Number of Named Selections

In an assembly or a part with a large number of named selections, close to about a thousand, NX might close unexpectedly when you try to add more than 10 named selections at a time in the list. To avoid the problem, add few named selections at a time, close the named selection manager and save the part. This problem does not affect a list of named selections less than about a thousand, and the workaround is not needed for such cases.

Length Unit

The Ansys Mechanical application automatically sets the length unit in the part or assembly to match the unit saved in the NX file. No adjustment of length unit is necessary or possible.

Material Property Values

There are known irregularities in material property values passed from NX to Ansys Workbench. Validate all material data prior to solving.

Missing BSpline Geometries in DesignModeler

Some solids, surfaces and lines containing Bspline geometries may not import correctly into the DesignModeler application, but should import correctly in the Ansys Mechanical application. Since these Bspline geometries do not conform to the Parasolid standard, they do not import correctly into the DesignModeler application. Newer geometries containing Parasolid-friendly Bsplines can be imported into the DesignModeler application without any problem.

Mixed Unit Support

It is recommended that all parts and sub-assemblies in a model should be in the same unit system. Models with mixed units will be imported into Ansys applications with all parts scaled to the

assembly unit system. A warning will be issued during the import indicating the mixed units of the model.

Multibody Parts

For multibody parts, the bodies are grouped together and placed under part node in the Ansys Mechanical application. If the multibody part is also multi-dimensional (that is, contains both surface and solid bodies), this may cause a meshing problem. Bodies with different dimensions in the same part cannot be meshed in Ansys Mechanical. An assembly, instead of a multi-dimensional multibody part, can solve the problem in such cases.

Named Selection Delete Problem with Wave Linked Features

Features containing wave linked geometries may not show the correct number of named selections after some of the named selections are deleted. For example, if an "Extracted Face" feature is created from a solid body and a named selection is created containing both the geometries, deleting that named selection may not delete it from the list. To fix the problem, close the Named Selection Manager (p. 25) and update the geometry using Edit> Feature> Playback or Tools> Update> Update for external change. The named selection manager should reflect the correct number of named selections after the update.

NX model import into Fluent on Linux may time out and fail

Work around this issue by modifying the `UGTserver.sh` script located under the `aisol/CADIntegration/linx64/` directory of the Ansys installation, inserting the following after the first line:

```
export UGII_BASE_DIR=your/nx/install/path/here
export SPLM_LICENSE_SERVER=your/license/server/here
```

Parameter Update Fails Instead of Displaying a Warning Message

The NX plug-in performs an undo operation when it encounters an error or a warning during a parametric update. The undo operation allows you to change the parameter back to a correct value. Fix the error or the warning message in NX before performing a parametric update on such a part. Alternatively, set an environment variable called UGII_UPDATE_FAIL_OPTION with a value “ACCEPT” or “ACCEPT_ALL” without the quotes and unset the variable once the parameter update is done. This will allow the import to work even if there is a warning or a failure.

Promoted and WAVE Linked Instances

If the instanced bodies are promoted, they will be treated as independent bodies. The algorithm to avoid duplicate copies of instanced bodies will skip promoted bodies. Thus, if there are assembly-level operations on the promoted instances, the modified bodies from those operations will get imported correctly. The same is true for WAVE linked instances as well.

Surface Body with Thickness

If your NX surface body model has a thickness defined and it does not get transferred to the Ansys Mechanical application, this could mean that you do not have an NX Scenario (Structural) license. Moreover, only the mid-surfaces created using the offset command can have thickness property and are imported correctly. Mid-surfaces created using face pair or user defined methods cannot have a thickness and cannot be imported. In addition to that, any thickness assigned on the surfaces inside the Advanced Simulation application cannot be imported in Ansys either.

Unique Parameters Name

When you transfer NX expressions/parameters into the Ansys Mechanical application, make sure that all feature parameter names are unique within a part and all non-feature parameter names are unique within an assembly. The non-feature parameters could be from one or more components of the assembly.

Limitations

Library Conflict

The following limitation is specific to the Ansys 2025 R1 and 2024 R2 Associative Geometry Interfaces to NX. There is a known library conflict with earlier releases of this Plug-In, which causes the 2025 R1 Plug-In to fail to load into NX when a previous release of the Plug-In is loaded into NX.

Use the following procedure to enable theese versions to be loaded side-by-side into NX.

1. Determine the path to the file specified by the environment variable UGII_CUSTOM_DIRECTORY_FILE. This can be done in one of two ways:
   - Open a Cmd window, and type “set UGII_CUSTOM_DIRECTORY_FILE”
   - Control Panel >> System >> Advanced System Settings >> Environment Variables. Take the value of the User variable unless it is only defined as a System Variable.
2. Open that file in an editor such as Notepad.
3. Reorder any lines containing <Ansys version>\ aisol\CADIntegration\Unigraphics\winx64 so that the newest (highest) Ansys versions come first in the file.
4. Save and close the file.

Parameters Placement – When a parameter's units are defined in the "Tools > Expressions" dialog in the formula field it will be processed as a dependent variable and placed in the Output list when promoted.

Simple version requirement– Siemens does not support multiple versions of the NX continuous release product line simultaneously installed on a single system. Such configurations may cause the Associative Geometry interface to fail.

Troubleshooting

See the NX Errors Related to CAD Integration (p. 130) section in CAD Integration Troubleshooting (p. 127) for detailed information.

Parasolid (*.x_t, *.xmt_txt, *.x_b, *.xmt_bin)

The application supports Parasolid files as a Reader (p. 133).

See Product & CAD Configuration Manager (p. 24) for usage information.

Linux: Configuring CAD Products> Using the CAD Configuration Manager

Windows: Configuring CAD Products> Using the CAD Configuration Manager

This is a stand-alone reader that does not require an installation of the Parasolid software or a Parasolid license.

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.x_t`, `*.xmt_txt`, `*.x_b`, `*.xmt_bin`,

Versions: 36.1 (Windows & Linux)

Table 18: Import Preference Support for Parasolid Geometry Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes 1
Named Selection Processing and prefix	Yes 1
Material Processing	No
Analysis Type	2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	Yes 1
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	No

1. Not supported in DesignModeler

Notations

The DesignModeler application allows neutral binary Parasolid files (x_b and xmt_bin) and text Parasolid files (x_t and xmt_txt) to be imported and exported.

Both text and neutral binary Parasolid files are platform independent.

Binary neutral Parasolid files (xmt_bin, x_b) are compressed but are not human readable.

Text Parasolid files are human readable but take up more space than their respective neutral binary versions.

The default Parasolid file setting for the DesignModeler application is text.

Faceting Limitation

Generalized (non-manifold) bodies from Parasolid will not have facets with smoothly matching edges. The model will appear to have gaps. However, this is a limitation only in the faceting capabilities from Parasolid and does not affect meshing or accuracy.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Length Unit

The Ansys Mechanical application automatically sets the length unit in the part or assembly to meters, which is the unit used internally by Parasolid to dimension solid parts. No adjustment of length unit is necessary or possible.

Part and Assemblies Files

Parasolid files containing parts or assemblies are supported by the Ansys Mechanical application. Parasolid files can use the extensions `.x_t` or `.xmt_txt` (text) and `.x_b` or `.xmt_bin` (binary neutral).

Body Instancing

If the **Import Using Instances** property is enabled, then body instances will be generated in Mechanical/Meshing for any Parasolid file with body instances.

Caveats and Known Issues

No caveats or known issues.

Troubleshooting

See the Parasolid Errors Related to CAD Integration (p. 130) section in CAD Integration Troubleshooting (p. 127) for detailed information.

Revit (*.rvt, *.rfa)

The interface works in a Plug-in (p. 133) mode.

This interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.rvt`) and Assembly (`*.rfa`)

Versions: 2024

Table 19: Import Preference Support for Revit Geometry Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	No
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	No
Named Selection Processing and prefix	No
Material Processing	No
Analysis Type	3D - Yes 2D - No
Associativity	No
Coordinate Systems	No
Work Points	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No

Decompose Disjoint Surfaces	No
Mixed Import Resolution	No
Clean Geometry On Import	No
Stitch Surfaces On Import	No
Import Facet Quality	No

Notations

Selective Update

This interface does not support the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Caveats and Known Issues

The Revit Plug-in has the following limitations:

Only 3D Geometry Is Supported.

Only 3D geometry from CAD formats link is supported.

No Support for Wires and Mesh.

Only 3D geometry is supported. Wires and Mesh are not supported.

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Geometry Interface is Beta

This Revit Geometry Interface is a Beta feature and untested and may result in unpredictable behavior both within Revit and Ansys applications consuming the transferred geometry.

Rhinoceros (*.3dm)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader, which does not require that Rhinoceros be installed to the system.
- No CAD associativity support.
- No CAD Parameters support.

The interface is always configured by the Ansys installation.

Support

At the time of release, detailed version support information for the Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `Assemblies and Parts (*.3dm)`

Versions: Rhino (3DM) 8

Table 20: Import Preference Support for Rhinoceros Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	No
Named Selection Processing and prefix	No
Material Processing	No
Analysis Type	3D - Yes 2D - surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	No

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

You should specify the length unit of the parts retrieved from the Rhinoceros file in the Details View. You can verify length units by checking the details of the Geometry or Part, which shows the part's bounding box size. You should verify dimensions before solving. See Solving Overview.

Hidden Parts

Parts that are hidden or suppressed in Rhinoceros are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Caveats and Known Issues

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

No Rhinoceros errors related to CAD Integration appear in CAD Integration Troubleshooting (p. 127).

Solid Edge (*.par, *.asm, *.psm, *.pwd)

For more information, see:

Solid Edge Reader (*.par, *.asm, *.psm)

Solid Edge Associative Geometry Interface (*.par, *.asm, *.psm, *.pwd)

Solid Edge Reader (*.par, *.asm, *.psm)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the Solid Edge system be installed.
- No CAD associativity support.
- No CAD Parameter support.

The interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.par`), Assembly (`*.asm`), and Sheet Metal (`*.psm`).

Versions: V1 – 2024 (Windows)

Table 21: Import Preference Support for Solid Edge Reader Interface

Import Solid Bodies	Yes
Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color
Named Selection Processing and prefix	Yes - Color
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	Yes
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden parts

Parts that are hidden or suppressed in Solid Edge are skipped automatically by this interface.

Import CAD Color as an Attribute or Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Caveats and Known Issues

The Solid Edge Reader has the following limitations:

- **Coordinate Systems.** Translation of coordinate systems (WCS) from Solid Edge is not supported.
- **Work Planes.** Translation of work planes from Solid Edge is not supported.
- **Attributes.** Translation of User Defined Attributes, both Part level as well as Assembly level, is not supported for Solid Edge Reader.
- **Material Properties.** The Material Properties will always be in the MKS unit system irrespective of the Solid Edge file unit system. The Reader supports only part level material properties.
- **Layers.** Reading and translation of Layer information for any of the entities is not supported.
- **Colors.** Translation of assembly definition and assembly instance color is not translated.
- **Hidden Parts.** Instance level visibility is not honored. Invisible instances will always be translated.
- **Limited Support.** Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Troubleshooting

See the Solid Edge Errors Related to CAD Integration (p. 130) section in CAD Integration Troubleshooting (p. 127) for detailed information.

Solid Edge Associative Geometry Interface (*.par, *.asm, *.psm, *.pwd)

The interface works in both a plug-in and a Pseudo-Reader (p. 133) mode. Both part and assembly document import is supported.

Solid Edge based on registry entries recognizes the existence of the plug-in. After opening a document in Solid Edge, if the **Ansys (version specific)** menu is not displayed in the Solid Edge menu bar, check if **Ansys(version specific)** is listed in the **Available Add-Ins** list box of the **Add-in Manager** dialog box (**Applications> Add Ins> Add-In Manager**). If it is listed but not checked, check the box in front of it and click **OK**. If it is not listed, run the Product & CAD Configuration Manager (p. 24) with Solid Edge selected for configuration.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager in the Ansys, Inc. Windows Installation Guide.

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

The Ansys Solid Edge interface supports Solid Edge part, assembly and sheet metal, both in traditional and synchronous formats, and traditional weldment documents.

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.par`, `*.asm`, `*.psm`, `*.pwd`,

Table 22: Import Preference Support for Solid Edge Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes
Parameter Processing and prefix/suffix key	Yes
Attribute Processing and prefix	Yes Yes - including Colors (keyword Color)
Named Selection Processing and prefix	Yes Yes - including Colors (keyword Color)
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes; surface and line bodies on the xy plane may be imported
Associativity	Yes
Coordinate Systems	Yes
Work Points	No
Reader Save File	No

Instancing	Yes
Smart Update	Yes
Enclosure and Symmetry Processing	No
Mixed Import Resolution	Yes
Decompose Disjoint Geometry	No

Notations

Closed Surface

A closed surface body will be imported as a solid body since Solid Edge considers this body as a solid.

Interpart Copies

The interpart copies of a part should be hidden in the part level to avoid importing it in the assembly.

Line Body Types

The following feature types can be imported from Solid Edge into Ansys Workbench as line bodies:

- Key Point Curves
- Curve by Tables
- Project Curves
- Derived Curves
- Cross Curves
- Copy Construction Curves
- Contour Curves

Simplified Model

For a part that has a simplified model (if the model is displayed in Solid Edge), you will need to have the **Simplify** menu displayed to get the simplified model in the AnsysMechanical application.

Parameters

Parameters have limited precision associated with Solid Edge models. By default, Solid Edge only shows two digits of precision past the decimal point. Therefore when you input 41.012 for example, and refresh, the precision value will appear in Ansys Workbench to be 41.01 after the update completes. If you increase the display precision in Solid Edge, you will then see the more precise parameter values in Ansys Workbench.

Selection Filter Toolbar

A toolbar named "**Ansys Selection Filter**" is available in Solid Edge to pick entities like vertex, edge and face for Name Selection creation. This toolbar becomes active only when the Named Selection Manager (p. 25) dialog is open.

Using Named Selection

After the Named Selection dialog box is launched from the Ansys toolbar, the Ansys selection filter bar will be activated. First select a filter, for example face filter, then select the faces with the shift key pressed. Click the **Create** button to finish creating a Named Selection with the selected faces. To modify a Named Selection, start by selecting it from the list and highlight it by clicking the **Select** button. Shift + click to add to or right-click to remove from the selected faces. Click the **Replace** button to complete the modification.

Specific to Ansys DesignModeler

When importing a Solid Edge assembly, make sure that no two components use the same component name. This will result in the second component being displayed on top of the first.

Part documents should contain only one body, otherwise a duplicate set of parameters and variables may be imported.

When the attributes flag is on and the DDM prefix is specified, attributes are created for each entity to allow import of motion loads.

Specific to the Mechanical application

All multi-solid-body components created in Solid Edge will be transferred to the Ansys Mechanical application as a single part containing multiple bodies.

The Part length unit within the Ansys Mechanical application is meters independent of the unit system displayed in Solid Edge. The **Length Unit** displayed under **Details of Geometry** in the Ansys Mechanical application cannot be changed.

To import motion loads from Solid Edge models to the Ansys Mechanical application, you will need to use the motion load files generated from the same version of Solid Edge that you are running. If the load files were generated from a different version of Solid Edge, the loads will not be imported properly. When the attributes flag is on and the DDM prefix is specified, attributes are created for each entity to allow import of motion loads.

Caveats and Known Issues

Active Document

If more than one document is open in Solid Edge, the top-most document, which is the active document, will be picked by the interface as the active document.

Import Solid Edge Construction Objects Control

The Solid Edge geometry interface does not import construction objects by default. The import of construction objects can be enabled by the `ANS_SE_IMPORT_CONSTRUCTION_OBJECTS` environment variable. Possible values are:

0: Do not import construction objects

1: Import construction solids + surfaces + wires

2: Import construction solids + surfaces only

3: Import construction solids + wires only

4: Import construction surfaces + wires only

5: Import construction solids only

6: Import construction surfaces only

7: Import construction wires only

Note that this environment variable may get removed entirely or replaced by a more user-friendly variation in a future release.

Large Model

The import of large Solid Edge models with a maximum length over 1 kilometer cannot be imported due to Parasolid precision, that is, all parts of a body must lie inside the Parasolid size box. This box is 1000 units on each side and is centered at the origin. It represents the whole of the model space.

Multiple Named Selection Filter Menu After Changing Ansys Language

After language setting is changed in Ansys, multiple menu for the named selection may appear in the Solid Edge Ansys Toolbar. To fix the problem unconfigure the plug-in using Product & CAD Configuration Manager (p. 24) (CCM) and open Solid Edge files one of each type, such as part (*.prt), assembly (*.asm), sheet metal (*.psm) and weldment (*.pwd). Close Solid Edge and reconfigure using Product & CAD Configuration Manager (p. 24) to fix the issue.

Plug-In Availability Considerations

Due to the architecture of the CAD's Add-In manager, the Solid Edge geometry interface will automatically load into the Solid Edge sessions of all users when any administrative user has configured the geometry interface only to be available for their account. This is the case even when the user has not configured the geometry interface for himself/herself and a global configuration has not been performed to enable the plug-in to run for all users. When encountering such a state it may be possible to import geometry from an active Solid Edge session, but any attempts to attach or refresh geometry without Solid Edge running will fail.

When the Solid Edge plug-in is configured as part of a network install (see Using the CAD Configuration Manager) unconfiguring it on the local, client machine will (a) remove the Ansys menu from Solid Edge, and (b) disable the ability to import geometry from an active Solid Edge session. However, import of Solid Edge models when the CAD is not active may continue to work until the interface is unconfigured on the server machine.

Unlocked Parameters for Synchronous Features

The parameters created during synchronous modeling are not locked by default. As a result the geometry cannot be modified by changing the unlocked parameters. For the same reason, any parametric modification from Ansys would fail to generate. To fix this problem, hold the cursor on the unlocked parameter label in Solid Edge and click the lock icon on the dialog box that pops up. Alternatively, the lock status can be modified from the Solid Edge Tools > Variables list. After locking the parameter, the geometry can be modified by changing it and can be driven by parametric update from Ansys.

Solid Edge Part Name.

The Solid Edge plug-in uses the part name as the identifier for the associativity mechanism to work. Renaming the part after import may break the associativity. The part can be renamed in Solid Edge and refreshed to reflect the changes.

Troubleshooting

See the Solid Edge Errors Related to CAD Integration (p. 130) section in CAD Integration Troubleshooting (p. 127) for detailed information.

SOLIDWORKS (*.sldprt, *.sldasm)

For more information, see:

SOLIDWORKS Reader (*.sldprt, *.sldasm)

SOLIDWORKS Associative Geometry Interface (*.sldprt, *.sldasm)

SOLIDWORKS Reader (*.sldprt, *.sldasm)

The interface works in a Reader (p. 133) mode.

- This is a stand-alone reader which does not require that the SOLIDWORKS system be installed.
- No CAD associativity support.
- No CAD Parameter support.

The interface can be configured during Ansys installation or by using the Product & CAD Configuration Manager (p. 24).

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.sldprt`) and Assembly (`*.sldasm`)

Versions: 97 — 2024 (Windows)

Table 23: Import Preference Support for SOLIDWORKS Reader Interface

Import Solid Bodies	Yes

Import Surface Bodies	Yes
Import Line Bodies	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes - Color
Named Selection Processing and prefix	Yes - Color
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes

Notations

Selective Update

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden parts

Parts that are hidden or suppressed in SOLIDWORKS are skipped automatically by this interface.

Import CAD Color as Named Selections

To import an object's Color as Named Selections in Ansys Workbench, specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Caveats and Known Issues

No Support for a Color Attribute assigned to an Instance of a Part or Assembly.

Color information is read only when it is present in the B-rep data section in the .sldprt file.

Limited Support for Reading Units in Part or Assembly Files.

Reading unit information from SOLIDWORKS parts of assembly files is only supported for file version Sw2001 through the latest and currently supported SOLIDWORKS version. For earlier versions, the unit is assumed as meter. Also, unsupported units such as *feetandinch* and *angstrom* are assumed as meter.

Material Properties

The Reader translates material properties for part files from SOLIDWORKS version 2012. The Material Properties will always be in the MKS unit system irrespective of the SOLIDWORKS file unit system. The Reader supports only part level material properties.

No Support for Assembly Features.

Only the data in the B-rep section of a file is read.

No Support for Attributes Show, No-Show and Hidden for Versions Prior to Sw2004.

Attributes show, no-show and hidden are supported from version Sw 2004 through the latest and currently supported SOLIDWORKS version.

No Configuration Support for Assemblies and Sub-Assemblies for Versions Prior to Sw98.

Configuration support is present for version Sw98Plus through the latest and currently supported SOLIDWORKS version.

No Support for Writing Actual Instance Names

The reader does not supports actual instance names for separate parts and sub-assemblies in XML write.

Part Configuration Support.

SOLIDWORKS part files can contain different configurations for the same feature. Each configuration has separate geometry and topology data. The reader extracts the B-rep data stored in the ".SLDPRT" file. SOLIDWORKS stores B-rep data corresponding to each configuration in separate streams of the ".SLDPRT" file. To support reading of configurations, reader requires the B-rep data of that configuration. Typically, a SOLIDWORKS part file when re-saved in a higher version does not contain B-rep data for all configurations in the file. Hence, the reader cannot translate these files completely. In such cases, open the part in SOLIDWORKS, activate each configuration and save the part.

No Support for Translation of Hidden and Suppressed Bodies/Features in Part.

Reader does not support translation of hidden and suppressed bodies/features in part. Those features will simply be translated as they are ignoring their hidden or suppressed status in the part file.

No Support for Translation of Coordinate Systems.

No Support for Translation of Free Curves, Free Points or Work Planes.

No Support for User Defined Attribute (UDA) Defined on Configurations.

Limited Support

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

Limitations

Limited Support for Virtual Assemblies

The SOLIDWORKS reader does not support pure Virtual Assemblies. To import a Virtual Assembly then the supporting parts must also be saved as part files.

Troubleshooting

See the SOLIDWORKS Errors Related to CAD Integration (p. 130) section in CAD Integration Troubleshooting (p. 127) for detailed information.

SOLIDWORKS Associative Geometry Interface (*.sldprt, *.sldasm)

The interface works in both a Plug-in (p. 133) and a Pseudo-Reader (p. 133) mode.

The existence of the Plug-in (p. 133) is recognized by SOLIDWORKS based on registry entries. After opening a document in SOLIDWORKS, if the **Ansys** (version specific) menu is not displayed in the SOLIDWORKS menu bar, check if **Ansys** (version specific) is listed in the **Available Add-Ins** list box of the **Add-in Manager** dialog box (**Tools> Add Ins**).

If listed: but not checked, check the box in front of it and click **OK**.

If not listed: run the Product & CAD Configuration Manager (p. 24) with SOLIDWORKS selected for configuration (Start->Workbench->Utilities->Product & CAD Configuration Manager (p. 24)). Select the Ansys /Workbench product and then the SOLIDWORKS interface. Click the **Next** button and then **Configure Selected Plug-Ins**. The output window should indicate success. (This requires administrative privileges on the machine.) If it fails or trouble persists, re-install the SOLIDWORKS plug-in component or contact your technical support service representative.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Windows (p. 6) platform is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.sldprt`) and Assembly (`*.sldasm`)

Table 24: Import Preference Support for SOLIDWORKS Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes - 3D spline curves, helix, curve in file, composite curve, ref curve, and imported curve
Parameter Processing and prefix/suffix key	Yes - In addition to dimension names, global variables are also processed for parameter. The same filter logic is used to determine which parameters are imported. Parameters that are driven are NOT imported.
Attribute Processing and prefix	Yes - From programmatically created attributes. Motion loads are imported using the preference and the corresponding third party attribute prefix.
	Yes - including Colors (keyword Color)
Named Selection Processing and prefix	Yes - From AnsysNamed Selection Manager (p. 25) and programmatically created attributes
	Yes - including Colors (keyword Color)
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes; surface and line bodies on the xy plane may be imported
Associativity	Yes - Some data is stored in the SOLIDWORKS model file and must be saved AFTER attach to maintain persistence. The Reader Save File option will need to be turned ON if imported in "reader" mode or you will need to save the file in an active SOLIDWORKS session otherwise. The associative mechanism consumes a good part of the attach/update time when importing models from SOLIDWORKS. If associativity is *not* necessary turning this option off will typically speed up the import/refresh time substantially, especially on larger models. The use of named selections can also be used as a persistence mechanism and does not require the associativity option be turned on.
Coordinate Systems	Yes - Part and assembly local coordinate systems
Work Points	Yes - Part and assembly local work points
Reader Save File	Yes
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed import Resolution	Yes - For parts that include both Solid and Surface bodies AND the Import Solid and Import Surface preferences are set to Yes. The Solid and Line option and the Solid, Surface and Line option are not supported.

Decompose Disjoint Geometry	No

Notations

The SOLIDWORKS geometry interface also fully supports the Selective Update feature.

Ansys Workbench automatically locks the length unit in the part or assembly to meters, which is the unit used internally by SOLIDWORKS. No adjustment of length unit is necessary or possible. The Ansys Mechanical application user can change the unit system for display of the Ansys Mechanical application data.

Parts that are hidden or suppressed in SOLIDWORKS are skipped automatically by the geometry interface.

If more than one document is open in SOLIDWORKS, the top-most document, which is the active document, will be processed by the interface.

Lightweight components (marked with a feather icon in the feature tree) of a SOLIDWORKS assembly must be set to resolved prior to attaching into the Ansys Mechanical application.

For parameters that include a unit the Workbench associative geometry interface will import just the numeric value and not the unit. When parameters are pushed back to SOLIDWORKS, it will replace the number and preserve the unit string. For example: MY_Param = 12mm in SW will import 12. If the value is changed to 10 in WB and updated the equation should be updated to MY_Param = 10mm.

Caveats and Known Issues

There is a limitation imposed by SOLIDWORKS in relation to geometry and the API processing. If a sketch is revolved 180 degrees, the faces generated on either portion of the revolution are identified as the same. However if the revolution angle is changed, they now become different faces. One retains the original identification and the second a new one. This creates an associativity break if the angle of revolution is modified to or from 180 degrees. If this situation arises you will need to reapply loads and/or boundary conditions.

Databases in which loads and/or boundary conditions are attached to surface body part entities are subject to a loss of associativity if created and saved prior to release 12.0. Once the loads are reattached, associativity should be maintained.

Unsaved SOLIDWORKS geometry files are not supported for import.

Loading Multiple Versions

In some cases, the SOLIDWORKS Associative Geometry Interface will fail to load into SOLIDWORKS. The work-around is to copy the following runtime libraries from %AWP_ROOT<ver>\aisol\bin\winx64 to %AWP_ROOT<ver>\aisol\CADIntegration\SolidWorks\winx64:

- libcrypto-3-x64.dll
- libssl-3-x64.dll

- Xalan-C_1_12.dll
- XalanMessages_1_12.dll
- xerces-c_3_2.dll
- zlib-1.2.13.dll
- zlib1.dll

Plug-In Availability Considerations

A scenario can occur where Attach/Refresh Geometry from active SOLIDWORKS session works correctly, however Attach/Refresh Geometry without SOLIDWORKS running fails with "Geometry Interface Not Found."

This can occur if you run the Product & CAD Configuration Manager (p. 24) as Administrator, and select Configuration Actions apply to: Current User.

The architecture of the CAD's Add-In manager will however display the Ansys plug-in for All Users of the machine, even if the "All Users" configuration has not been performed to enable the plug-in to run for all users. If this is done, the current user configuration of Ansys Workbench is not made aware of the SOLIDWORKS file types, and the interface is not found.

Workaround: Run the Product & CAD Configuration Manager (p. 24) as Administrator, select "All Users", and configure the SOLIDWORKS plug-in.

Limitations

Global Parameters

Global parameters that contain only numeric data in the equation field are processed. Equations which include units are skipped. Global Variables, defined at the part level, do not come through when an assembly file containing the part is imported.

Troubleshooting

See the SOLIDWORKS Errors Related to CAD Integration (p. 130) section in CAD Integration Troubleshooting (p. 127) for detailed information.

SpaceClaim (*.scdoc, *.scdocx)

Units Usage

The units defined in the Ansys Workbench project schematic's Unit System Dialog are not used in SpaceClaim. The imported model unit system for SpaceClaim is meters unless small unit support is turned on, and then the expected unit system of import is millimeters.

Importing SpaceClaim Files into Discovery

Discovery can import the SpaceClaim file format (*.scdoc, *.scdocx) regardless of the SpaceClaim version the file was saved in.

Also see:

SpaceClaim Associative Plug-In (p. 115)
CAD Integration SpaceClaim (p. 119)
Discovery (*.dsco) (p. 80)

SpaceClaim Associative Plug-In

The SpaceClaim geometry interface supports SpaceClaim Engineer, and with SpaceClaim Direct Modeler on the Windows 64-bit operating system.

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: Part (`*.scdoc`)

Table 25: Import Preference Support for SpaceClaim Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes
Parameter Processing and prefix/suffix key	Yes
Attribute Processing and prefix	No
Named Selection Processing and prefix	Yes
Material Processing	Yes
Analysis Type	3D - Yes 2D - Yes
Associativity	Yes
Coordinate Systems	Yes
Work Points	No
Reader Save File	No
Instancing	Yes — Workbench supports instances among parts that are represented by proper rigid transformations; that is, transformations which preserve the size, shape, and

	handedness of the part. Reflective or mirrored transformations are not supported.
Smart Update	Yes
Enclosure and Symmetry Processing	Yes (See Caveats and Known Issues section below)
Mixed Import Resolution	Yes
Decompose Disjoint Geometry	No

Caveats and Known Issues

Enclosure and Symmetry Processing

In SpaceClaim you can create volume extraction bodies and enclosure bodies. These are transferred independent of the "Enclosure and Symmetry Processing" setting. If you do not want them to be transferred, you can hide them in SpaceClaim.

Instancing Support

While SpaceClaim supports transformations of reflected or mirrored geometry, Ansys Workbench does not support it. The user is advised to remove the instancing relationships among reflected components in SpaceClaim before the geometry is imported in Ansys Workbench.

STEP (*.stp, *.step)

See Product & CAD Configuration Manager (p. 24) for usage information.

For detailed installation information about the Product & CAD Configuration Manager (p. 24), see:

Linux: Configuring CAD Products> Using the CAD Configuration Manager

Windows: Configuring CAD Products> Using the CAD Configuration Manager

Support

At the time of release, detailed version support information for the Linux (p. 4) and Windows (p. 6) platforms is accessible via Geometry Interface Support (p. 3).

For information about post-release CAD system compatibility with Ansys Workbench, see the Platform Support section of the Ansys Website.

Document import supported by interface: `*.stp`, `*.step`

Table 26: Import Preference Support for STEP Geometry Interface

Import Solids	Yes
Import Surfaces	Yes
Import Lines	Yes
Parameter Processing and prefix/suffix key	No
Attribute Processing and prefix	Yes

Named Selection Processing and prefix	Yes - Color and Layer (No for DesignModeler)
Material Processing	No
Analysis Type	2D - Yes- surface and line bodies on the xy plane may be imported
Associativity	No
Coordinate Systems	No
Work Points	No
Reader Save File	No
Instancing	No
Smart Update	No
Enclosure and Symmetry Processing	No
Mixed Import Resolution	No
Decompose Disjoint Geometry	Yes (No for DesignModeler)
Clean Geometry On Import	Yes (No for DesignModeler)
Stitch Surfaces On Import	Yes (No for DesignModeler)
Import Facet Quality	Yes

Notations

Specific to non-DesignModeler products

Closed faces and hollow solids from an STEP file are transferred as full solids in the Ansys Mechanical application.

Selective Update (Windows)

This interface fully supports the Selective Update feature.

Length Unit

The length unit specified in the part or assembly is automatically read from the geometry file and is transferred into Ansys Workbench. The Length Unit field in the Details will be read-only. If a unit system is not detected in the geometry file, the Length Unit field in the Details will be active, allowing the Length Unit to be specified.

Hidden parts

Parts that are hidden or suppressed in CATIA V5 are skipped automatically by this interface.

Import CAD Color as an Attribute of Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Color" or a list of specific Colors delimited by a semicolon (for example: "Color:255.0.0; Color:255.255.0").

Import CAD Layer as an Attribute of Named Selection

- **Attribute**: specify the "Attributes" option and set the "Attributes Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3").
- **Named Selection**: specify the "Named Selections" option and set the "Named Selections Key" to "Layer" or a list of specific Layers delimited by a semicolon (for example: "Layer:0;Layer:1;Layer:3")

Specific to Ansys DesignModeler

You cannot import STEP files containing models with bounding box greater than 500 cubic meter in any direction from world origin even with "Enable Large Model Support" option turned on. For STEP imports to be successful the model must fit inside the base 1 km^3 size box.

The STEP (STandard for the Exchange of Product model data) reader will both read and write model data to and from the STEP format. It is important to note that the STEP format does not store model data in the same way as the DesignModeler application. STEP format stores surface data, which upon import into the DesignModeler application is stitched together to form bodies. In some rare cases, the DesignModeler application model exported to STEP format may not produce the exact same geometry when imported again into the DesignModeler application.

Caveats and Known Issues

For STEP assembly models containing any assembly level parts and root level parts of the same name, the root level parts with those duplicate names fail to import, unless they are imported into DesignModeler or SpaceClaim Direct Modeler first.

Non-Commercial, Education Edition, Alpha, Beta, and PR versions (even of supported official versions) are not supported.

CAD Integration SpaceClaim

Ansys SpaceClaim Direct Modeler (SCDM) is available in Ansys Workbench. Unlike Ansys DesignModeler which is a history-based parametric application, SpaceClaim Direct Modeler is a direct modeling application. Access and use of SpaceClaim Direct Modeler requires you have an existing SpaceClaim Direct Modeler license.

SpaceClaim topics:

Import Procedure

You can select your preferred geometry editor in the Options panel via the Tools menu in Ansys Workbench. Select Geometry Import.

SpaceClaim Direct Modeler allows you to import a clean instance of a 3D model from a CAD system without history-based parameters. However you cannot conduct a parametric study using the same parameters that are defined in the native CAD model, and after an analysis, update the model through the same set of parameters automatically.

Within the Ansys Workbench environment, four options exist for SpaceClaim depending when the work flow commences.

- Importing External CAD Files into SpaceClaim Direct Modeler (SCDM) using Ansys Workbench (p. 120)
- Importing SpaceClaim Geometry into DesignModeler (p. 120)
- Starting SpaceClaim before Ansys Workbench (p. 120)
- Starting Ansys Workbench before SpaceClaim (p. 121)

Importing External CAD Files into SpaceClaim Direct Modeler (SCDM) using Ansys Workbench

1. Start Ansys Workbench
2. Create empty geometry cell.
3. Right-click the geometry cell and select Import Geometry->Browse. Specify the CAD file name in the file open dialog.
4. Make sure the geometry file name is set in the geometry cell properties.
5. Right-click the geometry cell and select "Edit Geometry in SpaceClaim"

Importing SpaceClaim Geometry into DesignModeler

Note:

You do not need to close a running instance of SpaceClaim Direct Modeler to import SpaceClaim geometry into Ansys DesignModeler.

1. Launch "SpaceClaim Direct Modeler":
2. Open an existing .scdoc file (or) create geometry in SpaceClaim and save into a `.scdoc` file.
3. Launch Ansys Workbench.
4. Create a new Geometry system in Project Schematic.
5. Launch Ansys DesignModeler using the right mouse button on the Geometry cell. Select **New DesignModeler Geometry**.
6. Inside Ansys DesignModeler, select the menu item **File->Attach to Active CAD Geometry**. The "Source" property of Attach feature should be set to the `.scdoc` file.
7. Click the **Generate** button. The SpaceClaim geometry is imported into Ansys DesignModeler.

The following steps can be used to bring SpaceClaim geometry in to Ansys DesignModeler using the SpaceClaim Plug-in (p. 133). The following steps will not use SpaceClaim Add-in.

Starting SpaceClaim before Ansys Workbench

In this scenario, Ansys Workbench is started using the SpaceClaim menu item, either Ansys SpaceClaim Direct Modeler or SpaceClaim Engineer.

Using Ansys SpaceClaim Direct Modeler

If geometry in Ansys SpaceClaim Direct Modeler is not saved to a `.scdoc` file:

1. Start Ansys SpaceClaim Direct Modeler and create geometry, but do not save.

2. Starting Ansys Workbench using the 2025 R1 button in the Prepare tab will create a geometry system in the project schematic.

3. The "geometry file name" property of the geometry cell will be empty.

4. The "CAD Plug-in" property will be set to "SpaceClaim[]" if geometry in SpaceClaim Direct Modeler is not saved in to a `.scdoc` file.

5. As soon as Ansys Workbench is started, the SpaceClaim Direct Modeler editor will behave in add-in mode and you can transfer geometry to Mesh/Mechanical.

Using Ansys SpaceClaim Direct Modeler

For saved `.scdoc` file:

1. If geometry in Ansys SpaceClaim Direct Modeler is saved to a `.scdoc` file before starting Ansys Workbench, then the "geometry in name" property is set to the saved `.scdoc` file name.

2. As soon as Ansys Workbench is started, the SpaceClaim Direct Modeler editor will behave in add-in mode and you can transfer geometry to Mesh/Mechanical.

Using SpaceClaim Engineer

you should save the geometry in SpaceClaim Engineer to a `.scdoc` file before starting Ansys Workbench.

For a saved `scdoc` file:

1. If the geometry in SpaceClaim Engineer is saved into a `scdoc` file before starting Ansys Workbench, then "geometry file name" property is set to the saved `scdoc` file name and "CAD Plug-in" property as "SpaceClaim[]".

2. You can transfer geometry to mesh/mechanical.

Starting Ansys Workbench before SpaceClaim

In this scenario, SpaceClaim is started using the Ansys Workbench menu item.

1. Start Ansys Workbench

2. Create empty geometry cell.

3. Start SpaceClaim Direct Modeler and create geometry.

4. If you do not save the geometry in to `.scdoc` file, user could not attach geometry.

5. To attach the geometry, save the geometry to a `.scdoc` file.

6. If you attach a saved `.scdoc` file in the geometry cell, the SpaceClaim Direct Modeler editor will behave in add-in mode.

Importing SpaceClaim Geometry into Mechanical APDL

1. Start Ansys Workbench.
2. Create an empty geometry cell.
3. Right-click the geometry cell and select Import Geometry -> Browse to assign a `.scdoc` file or external CAD file.
4. Edit the geometry in SpaceClaim Direct modeler if required.
5. Establish a downstream connection to a Mechanical APDL cell.
6. Refresh the Mechanical APDL cell to execute a transfer from SpaceClaim Direct Modeler as an exported `.anf` file.

Limitations

SpaceClaim support for prior versions of Workbench is limited to Ansys 15.0 or newer.

Related Topics:

SpaceClaim Side-by-side Configurations

SpaceClaim Pre-16.1 Installation Guidelines

Ansys SpaceClaim Direct Modeler (SCDM) Behavior in the Project Schematic

SpaceClaim Side-by-side Configurations

Ansys Workbench 17.0 now allows multiple versions of SpaceClaim products to be installed on the same machine. The Product & CAD Configuration Manager (p. 24) utility offers advanced support for configuring how SpaceClaim links with Ansys Workbench. Ansys Workbench 17.0 supports two forms of side-by-side configurations for SpaceClaim products.

Ansys SpaceClaim Direct Modeler Side-by-side

Ansys SpaceClaim Direct Modeler was integrated into the Ansys Workbench installation starting with version 16.0. Ansys Workbench's side-by-side support now includes the Ansys SpaceClaim Direct Modeler component.

The Ansys SpaceClaim Direct Modeler included in the 17.0 installation can only be configured to link with the 17.0 Workbench components. Links to prior versions of Workbench are no longer supported.

SpaceClaim Engineer Side-by-side

SpaceClaim Engineer installations can be installed side-by-side with Ansys SpaceClaim Direct Modeler. The Product & CAD Configuration Manager (p. 24) provides the ability to configure the SpaceClaim link to Ansys Workbench 17.0.

The Product & CAD Configuration Manager (p. 24) utility will present the configuration options according to the supported versions it detects. Support for configuring SpaceClaim Engineer will only be visible

in the Product & CAD Configuration Manager (p. 24) utility if a valid Ansys Workbench 17.0 connection plugin is detected within its installation.

Note:

Only one SpaceClaim product can be configured with Ansys Workbench at a time.

Note:

If SpaceClaim Engineer is installed prior to the Ansys Workbench installation and the Product & CAD Configuration Manager (p. 24) utility can detect a supported connection plugin it will be automatically configured to be active.

SpaceClaim Pre-16.1 Installation Guidelines

Ansys SpaceClaim Direct Modeler is now part of the Ansys Workbench installation and requires configuration changes when versions prior to Workbench 16.1 or other SpaceClaim products (standalone Ansys SpaceClaim Direct Modeler, SpaceClaim Engineering) are installed or will be installed on the same machine.

SpaceClaim Products Installed Prior to Workbench 16.0 Installation

Ansys SpaceClaim Direct Modeler will not be configured during installation of Workbench if a previous installed version of SpaceClaim Direct Modeler or SpaceClaim Engineering is detected.

If desired, Ansys SpaceClaim Direct Modeler can be configured manually, following the removal of previously installed versions.

To configure manually, execute the AnsConfigSpaceClaim.exe from the following Ansys installation location: <INSTALL_PATH>\common files\configs\winx64\config\

Linking Workbench 15.0 with the SpaceClaim 16.0 Installation

The Ansys SpaceClaim Direct Modeler included in the 16.0 installation can be configured to interface with Workbench 15.0 if no other SpaceClaim products are installed. To establish the link, execute the SCDMConnect.exe from the following Ansys installation location: <INSTALL_PATH>\scdm\

Note:

Ansys SpaceClaim Direct Modeler 16.1 cannot be linked to prior versions of Ansys Workbench.

SpaceClaim Products Installed After Workbench 16.0 Installation

All SpaceClaim products released with and after the Workbench 16.0 release have installation support that will automatically unconfigure the 16.0 version of SpaceClaim Direct Modeler. This will override the Workbench 16.0 installation and allow the SpaceClaim product to assume the linked connection with Workbench.

Known Issues and Limitations

- The 16.0 version of SpaceClaim Direct Modeler is not compatible with Workbench versions prior to 15.0. Use of SpaceClaim Direct Modeler with Workbench versions prior to 15.0 requires SpaceClaim Direct Modeler 2014 SP2.
- Installing SpaceClaim products released prior to Workbench 16.0 is not supported. Doing so will result in unexpected behaviors. The 16.0 version of SpaceClaim Direct Modeler can be disabled to allow prior versions of SpaceClaim products to assume the linked connection to Workbench by executing AnsUnconfigSpaceClaim.exe from the following Ansys installation location: <INSTALL_PATH>\common files\configs\winx64\unconfig\

Note:

Once the 16.0 version of Ansys SpaceClaim Direct Modeler has been unconfigured, prior SpaceClaim product versions can be installed/re-installed.

Note:

All SpaceClaim Engineering products released with or after Workbench 16.0 have modified their installations to automatically unconfigure the 16.0 version of Ansys SpaceClaim Direct Modeler if it is detected.

Ansys SpaceClaim Direct Modeler (SCDM) Behavior in the Project Schematic

Within the Ansys Workbench help there is detailed information about using Ansys SpaceClaim Direct Modeler in the Ansys Workbench Schematic at:

Configuring Ansys Workbench> Setting Ansys Workbench Options> Geometry Import

Systems> Component Systems> Geometry

See the Workbench User's Guide for complete information about the Ansys Workbench interface.

Note:

A geometry cell can be edited by one editor at a time (either Ansys DesignModeler or Ansys SpaceClaim Direct Modeler).

CAD Integration Frequently Asked Questions

See the Geometry Interfaces section via the Products tab of the Ansys website for a list of On Demand webinars related to CAD integration.

The Ansys Knowledge Resources section of the Ansys customer site includes a database of frequently asked questions for such general topics as CAD Connections. Listed under the Online Support heading on the Ansys customer site, the Ansys Knowledge Resources section also includes best practices and examples.

CAD Integration Troubleshooting

Categorized below by CAD systems is a list of error messages and the cause of the error. The Ansys Knowledge Base (accessible via the Ansys customer site) allows you to perform solution and defect searches. You must register and create a password to access the Ansys customer site via the Ansys website (www.ansys.com).

Solutions Search: The Ansys Solutions Search allows you to search the Ansys Knowledge Base for a wide variety of technical topics. The Knowledge Base is continuously updated by the support organization with new solutions, best practices, examples, etc.

Defects Search: The Ansys Defect Search allows you to search the help defects files for selected Ansys, Inc. products.

To view the Ansys Knowledge Resources, navigate to Online Support> Knowledge Resources on the Ansys customer site.

General Errors Related to CAD Integration

For further clarification of errors listed below, contact Ansys Technical Support. More information about Ansys Technical Support and Technology Enhancements and Customer Support (TECS) is available via the Ansys customer site. You must register and create a password to access the Ansys customer site via the Ansys website (www.ansys.com).

Error Message	Cause of Error
The required license is unavailable.	Plug-in license was not able to be checked out.
Assemblies not licensed.	User attempting to attach an assembly model, but did not purchase or properly configure assembly licensing.
Cannot open part file.	If call to CAD to open model file fails this message is displayed.
Unable to retrieve part.	Part model is wireframe model in UG.
No model is currently active.	The CAD system has no model open.

Error Message	Cause of Error
Unable to acquire part data.	Query of CAD part data returned failure message.
Assembly has no parts.	An assembly model has no parts or all have been suppressed.
Unknown entity.	CAD identifies model element as a type unknown to plug-in.
Registration access error.	Attempt to read information for system registry returned error.
Plug-In not found.	An attempt to use a plug-in that does not exist.
Failed to load String Table.	Localized string table was not able to be loaded, may not exist. Error is not fatal, will cause messages to appear as somewhat cryptic.
Unable to activate document.	Unable to make identified model the active model.
There are no active parts in the model.	A part model has had the part suppressed.
Regeneration failure.	During an update, when changing a parameter value if the CAD is unable to update with the value set passed it this message will be displayed. When possible, the model is changed back to its previous state.
Attach failed.	General failure that indicates the attach was unable to be completed.
Refresh failed.	General failure that indicates the attach was unable to be updated.
No active document.	CAD system running with no model opened.
Unable to access the selected file.	The file is either protected against reading or does not exist in the specified location. Note that Creo Parametric users may not use models specified with a UNC path.
A Fatal Exception was caught.	An unexpected error occurred within the plug-in. Report issue to TECS.

CAD Geometry Fails to Import

If you attempt to import geometry and the Plug-in (p. 133) or Reader (p. 133) fails to load properly or the geometry fails to load, or if DesignModeler fails to start, the CAD configuration may not have been completed properly during the Ansys Workbench installation process. Although the product installation and CAD configuration steps can be done as a non-administrative user, administrative rights are required to fully configure several Ansys CAD products, including the Ansys Workbench Plug-Ins for Autodesk Inventor (*.ipt, *.iam) (p. 40), Creo Elements/Direct Modeling (p. 59), Solid Edge, and SOLIDWORKS. NX in Reader (p. 133) mode also requires administrative rights to configure. If these products were not configured as administrator, the registration may not have completed properly and you could see CAD-related errors. See Using the CAD Configuration Manager in the Ansys, Inc. Windows Installation Guide for detailed information on configuring these CAD products properly.

If you ran a silent prerequisite installation, you may need to reboot your machine to complete the prerequisite installation before you can successfully import a geometry.

ACIS Errors Related to CAD Integration

Error Message	Cause of Error
Unable to initialize (ACIS or Parasolid) libraries.	Current file structure has missing or erroneous elements.
File does not exist.	Entered file does not exist.
Failed to read the (ACIS or Parasolid) file.	File is corrupt.
No valid bodies found in the file.	No valid geometry found in file to import.

Autodesk Inventor Errors Related to CAD Integration

Error Message	Cause of Error
Failed to get reference key.	Unable to get reference data from Autodesk Inventor (*.ipt, *.iam) (p. 40). Not a fatal error. The attach process continues, but an associative relation during update cannot be guaranteed.
Attach failed.	Attach aborted.

CATIA Errors Related to CAD Integration

Error Message	Cause of Error
System Error	Current file structure has missing or erroneous elements.
File is not a CATIA export file	The Ansys Mechanical application can import only CATIA export type files, the entered file is not of that type.

Creo Parametric Errors Related to CAD Integration

Error Message	Cause of Error
The CAD model regenerated successfully, but some features are unattached.	One or more Creo Parametric model feature became unattached either prior to or as result of parametric change and regeneration initiated by Workbench Plug-In.
Unable to select one or more items from selection, possible corrupted WB parameters within Pro/E model.	The Ansys Workbench Plug-In stores named selection data as Creo Parametric Model Parameters with prefix WBNS or WBAC. When this message is displayed the Named Selection Manager (p. 25) likely failed to complete the specified add, remove or rename operation because one of these parameters was modified or deleted.
Unable to delete one or more items from selection, possible corrupted WB parameters within Pro/E model.	
Unable to rename one or more items from selection, possible corrupted WB parameters within Pro/E model.	
Cannot activate different model with same name as model in session.	User is attempting to activate a model of the same name and different path when attaching models in the Ansys Mechanical application.

Error Message	Cause of Error
Entered non-integer value for XXX Truncating value.	When changing the value of a parameter in the Ansys Mechanical application, if the originally defined parameter is of integer type and you enter a non-integer value this message will be displayed as a reminder for future updates.
Warning: Pro/E Asm does not use consistent unit system.	A component part does not have the same unit system as the assembly. This is a requirement as indicated in the documentation. The component name is visible in the progress window at the time this error occurs. You are advised to terminate the attach (by clicking **Cancel** in the progress window) and returning to Creo Parametric to bring all components of the model to the same unit system. If you wish, you may allow the attach to continue to its completion making note of all components that generate this warning, then return to Creo Parametric to make changes. In some instances only the assembly must be modified instead of the parts. If allowed to import entirely, the model is likely to have graphics and selection problems. These will be remedied when the model is imported after changes are made to the Creo Parametric model.

NX Errors Related to CAD Integration

Error Message	Cause of Error
Could not lock NX.	NX is already locked by some other process
Could not unlock NX.	If the AnsysMechanical application did not successfully lock NX, this unlock error is also likely to be displayed as it does not have the right to unlock NX.

Parasolid Errors Related to CAD Integration

Error Message	Cause of Error
Schema access error.	Probably saved model in later version of Parasolid or schema directory is missing.

Solid Edge Errors Related to CAD Integration

Error Message	Cause of Error
Workbench Plug-In disabled.	Plug-in has been disabled and unable to be used for attach process.

SOLIDWORKS Errors Related to CAD Integration

Error Message	Cause of Error
Rebuild of Part Failed.	Regeneration failed on update with parameter changes.

Error Message	Cause of Error
The mechanical database contains unlicensed objects.	Lightweight components (marked with a feather icon in the feature tree) of a SOLIDWORKS assembly must be set to resolved prior to attaching into the Ansys Mechanical application.

CAD Integration Glossary

The following terms define nomenclature that is unique to Ansys, Inc. or specialized in meaning.

Associative Geometry Interface

The general term for all plug-ins and pseudo-readers.

Instances

When multiple copies of the same assembly-part are present in the CAD this preference allows time and memory savings by importing only the first instance of the part within the assembly and creating references to the first part for subsequent instances.

Reader

Does not require the CAD system.

Plug-in

Requires that the CAD system be running.

Pseudo-Reader

CAD system is started by Ansys Workbench in batch mode and shuts it down after attach/update is completed.

Additional Glossaries

Application-specific glossaries are available for the following.

- **Ansys, Inc. Installation and Licensing Guide:** Ansys Help System> Installation and Licensing Documentation>Ansys, Inc. Licensing Guide> Glossary
- **Ansys CFX:** Ansys Help System> CFX> CFX Reference Guide> Glossary
- **Ansys DesignModeler:** Ansys Help System> DesignModeler> Typical Usage> Glossary
- **Ansys Mechanical:** Ansys Help System> Mechanical> Appendices> Glossary of General Terms
- **Ansys FLUENT:** Ansys Help System> FLUENT> Getting Started Guide> Glossary of Terms
- **Ansys Workbench User's Guide:** Ansys Help System> Workbench> Ansys Workbench User Guide> Systems> Glossary

CAD Integration Updates

For updated CAD-related information, see the customer site section of the Ansys, Inc. website. For example, there you can access CAD special versions, standard versions, and archived versions. First-time users of the customer site must register to create a password. To view additional documentation information and late changes, navigate to Knowledge Resources> Online Documentation.

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