

A Guide to Geometry Import and Repair in ABAQUS/CAE Version 6.4

This page describes the following:

- [Using the Elysium direct translators to import parts from CAD systems.](#)
- [Using the Elysium translator plug-ins to import parts from CAD systems.](#)
- [A guide to importing problematic geometry into ABAQUS/CAE.](#)

Using the Elysium direct translators to import parts from CAD systems.

ABAQUS provides direct translators that allow you to import parts from the following file formats directly into ABAQUS/CAE:

- CATIA V4: Versions 4.1.9 through 4.2.4
- Parasolid: Version 7 through 14. A variety of CAD products can generate Parasolid-format parts, such as Unigraphics, SolidWorks, Solid Edge, FEMAP, and MSC.Patran.

For more information about the Elysium direct translators, contact your local ABAQUS sales office.

Using the Elysium translator plug-ins to import parts from CAD systems.

In addition, you can import a part into ABAQUS/CAE using the Elysium Neutral File format. ABAQUS provides a translator plug-in from Elysium Inc. that will generate a geometry file using the Elysium Neutral File format. The plug-in is available for the following products:

- Pro/ENGINEER Version 2001 and Wildfire
- I-DEAS Master Series Version 9 and Version 10
- CATIA V5 Release 9-11

For more information and to download the plug-ins, refer to the [Elysium plug-ins](#) web page.

A guide to importing problematic geometry into ABAQUS/CAE.

If the appropriate plug-in or direct translator is not available, ABAQUS/CAE allows you to import parts from CAD systems using industry-standard formats such as IGES and VDA-FS. Unfortunately, the industry standards are not rigid enough to guarantee that a part can be imported seamlessly. After you import a part, you may have to edit and repair the part to make it usable by ABAQUS/CAE.

These web pages provide a guide to help you import parts into ABAQUS/CAE and repair any geometry that is problematic. We will continue to update these web pages with the latest information available to us. In addition, more information on importing and repairing parts is available in the printed and online versions of the ABAQUS/CAE User's Manual and from the ABAQUS Online Support System.

The following topics are covered in this guide:

- [What's new in ABAQUS Version 6.4?](#)
- [My import failed. What should I do now?](#)
- [CAD system export options](#)
- [Frequently asked import and repair questions](#)

If you are using ABAQUS/CAE Version 6.3, you should refer to the following:
[The ABAQUS Version 6.3 geometry import and repair guide.](#)

What's new in ABAQUS Version 6.4?

New formats

You can now import a part from CATIA V5 into ABAQUS/CAE using an Elysium direct translator. A plug-in for CATIA V5 is available from Elysium Inc. that will write a geometry file using the Elysium Neutral File format. A reader is available for ABAQUS/CAE that will read the Elysium Neutral File generated by CATIA. For more information, contact your local ABAQUS sales office.

Miscellaneous changes

- The repair options now recognize the type of part being imported. As a result, ABAQUS/CAE presents only the options that will help you repair the part. In addition, ABAQUS/CAE automatically repairs a part during the import process when you import a part using the following formats.
 - CATIA V4
 - Parasolid
 - I-DEAS Elysium Neutral File
 - Pro/ENGINEER Elysium Neutral File
 - CATIA V5 Elysium Neutral File
- Two tools have been added to the Repair toolset that can help you repair imported and repaired parts.
 - **Repair face normals**
You can repair the face normals of shell and solid imported parts. In rare cases the Query toolset reports that the volume of an imported solid part is negative. The volume appears to be negative because ABAQUS has turned the solid inside out. The Repair face normals will

flip the normals and turn the solid right side out. An imported shell part can contain faces that have normals pointing in opposite directions. The Repair face normals tool will align all the normals on a shell part. If the face normals are already aligned, this tool will flip all the normals so that they remain aligned but point in the opposite direction.

- **Repair invalid edges**

In rare cases after you import a part, ABAQUS will report that all of its edges are invalid. The Repair invalid edges tool will try to repair the invalid edges by recomputing the data that define them. You should also use this tool if only invalid edges remain after automatically repairing a part.

- When you import a part from a third-party CAD system, the dimensions of the part or the units in which the dimensions are defined may not match the rest of your ABAQUS/CAE model. You can make a global change of an imported part's dimensions by scaling the part during the import process. You can choose from the following options:
 - Choose **No scale** to maintain the dimensions stored in the file.
 - Choose **Use transform from file, including scale** to read the scale factor, the rotation matrix, and the translation matrix from the file.
 - Choose **Multiply all dimensions by**, and enter a scale factor.
- In some cases when you import an IGES- or VDA-FS-format part and select the Stitch edges repair option, ABAQUS/CAE imports separate parts as a single part. If you toggle on the **Separate disconnected regions into parts** option and copy the imported part to a new part, ABAQUS/CAE will separate disconnected regions into separate parts.

Release Notes

For more information, see Chapter 5, "Geometry import and repair" in the ABAQUS Version 6.4 [Release Notes](#).

My import failed. What should I do now?

The following is a list of potential import problems that can be solved while working in ABAQUS/CAE. For information about problems that must be solved by the CAD system exporting the file, see [CAD system export options](#).

General problems

- [Import creates an invalid part](#)
- [Import creates an imprecise part](#)
- [Nothing is visible in the viewport](#)
- [Nothing is visible in the viewport after automated repair](#)

- [Mesh fails or has very distorted elements](#)
- [Part still has invalid geometry after automated repair](#)
- [Some faces of the part are visibly incorrect](#)
- [Some faces of the part are very small](#)


IGES-specific problems

- [IGES overview](#)
- [Import displays: "Failed to read file"](#)
- [Import displays: "This IGES file contains an entity of type 186 which sometimes cannot be read"](#)
- [Import displays: "File does not contain entities from which to build a part"](#)
- [There are unwanted visible gaps between some neighboring faces](#)

Import creates an invalid part

Possibility	Resolution
The file contains surfaces that are not stitched together into a B-rep solid body. All IGES parts exported without the MSBO option and all VDA-FS parts are invalid when imported into ABAQUS/CAE.	To highlight the invalid entities, select Tools -> Query and select Geometry validity from the Query dialog box that appears. If ABAQUS/CAE highlights the entire part or all of its vertices, none of the surfaces are connected together. Use the automated repair process to try and stitch the surfaces together and create a valid part. Use the following table for help with the options that appear after you select Automated repair .

Automated repair options	Result	Further Requirements
Select the following : <ul style="list-style-type: none"> • Convert to analytical representation • Stitch 	May result in an imprecise part.	You can continue to work with an imprecise part; however: <ul style="list-style-type: none"> • ABAQUS/CAE may not be able to mesh the part. • Some geometry operations may fail.

edges		
<p>Select the following:</p> <ul style="list-style-type: none"> • Convert to analytical representation • Stitch edges • Click Yes when ABAQUS/CAE asks: Do you want the invalid geometry deleted now? 	<p>ABAQUS/CAE may remove some invalid geometry; however, the part may still be imprecise.</p>	<p>Use the manual geometry repair tools to replace the invalid geometry that ABAQUS/CAE removed. For more information, see What can I do with the manual repair tools?</p>
<p>Select the following:</p> <ul style="list-style-type: none"> • Convert to analytical representation • Stitch edges • Convert to precise representation 	<p>Selecting all three options does not guarantee a valid and precise part. In addition, selecting the Convert to precise representation option will slow down the repair process significantly. You should select this option only in conjunction with the first two options.</p> <p>You can use the shaded render style </p> <p>tool to look at the effect of the automated repair process. A face will not shade correctly if it does not coincide with the underlying surface. In</p>	<p>Use the manual geometry repair tools to remove the shadowed faces and create new faces. If many faces are affected, try selecting a different Trim Curve Preference in the IGES Options dialog box, such as Always use 3D data.</p>

	addition, hidden lines will appear on an undulating face, giving a criss-cross appearance.	
<p>Select the following :</p> <ul style="list-style-type: none"> • Convert to analytical representation • Stitch edges • Convert to precise representation • Click Yes when ABAQUS/CAE asks: Do you want the invalid geometry deleted now? 	Selecting all three options does not guarantee a valid and precise part. Trying to make the part precise may corrupt the data that ABAQUS/CAE generates while stitching faces.	Try working with the imprecise part created with the first two repair options selected.

ABAQUS/CAE generates a file called [abaqus_repair_part.log](#) that reports the results of the automatic repair process. For more information, see [What does automatic repair do?](#)

Import creates an imprecise part

Possibility	Resolution
The file contains wires and surfaces that are stitched together, but the precision is not within the limits of	To highlight the imprecise entities, select Tools -> Query and select Geometry precision from the Query dialog box that

<p>ABAQUS/CAE's native precision. For more information, see What is a valid and precise part?</p>	<p>appears.</p> <p>If only a few regions of the part are imprecise, you can use the manual repair tools to remove the imprecise entities by removing faces and creating new faces. Removing imprecise entities will enable you to create partitions or additional shape features. An imprecise part is sufficient for creating triangular and tetrahedral meshes.</p>
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Nothing is visible in the viewport

Possibility	Resolution
<p>The bounding box is affecting the display of the part.</p>	<p>Try using the automated repair tools to make the part visible.</p> <p>Create a datum coordinate system at the origin, and use the view manipulation tools to zoom in to the coordinate system. The part should become visible if it is near the origin. If a region of the part is not visible, it may be because the view is a box with finite dimensions and this region is not within the box.</p>

Nothing is visible in the viewport after automated repair

Possibility	Resolution
<p>Automated repair deletes wires.</p>	<p>If the part is two-dimensional and all wires, import the part as a sketch.</p>
<p>Automated repair deforms the part to such an extent that it cannot be displayed.</p>	<p>Use only the first two automated repair options:</p> <ul style="list-style-type: none"> • Convert to analytical representaiion

	<ul style="list-style-type: none"> • Stitch edges
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Mesh fails or has very distorted elements

Possibility	Resolution
Very small faces or edges in part.	<p>Use the manual repair tools to merge the small face:</p> <ol style="list-style-type: none"> 1. From the Geometry Repair Tools, select Remove face to remove the unwanted face. 2. From the Geometry Repair Tools, select Automated repair with the Convert to analytical representation and Stitch edges options toggled on.
	<p>You can use the Mesh module to diagnose the part. Do the following to determine which faces fail to mesh:</p> <ol style="list-style-type: none"> 1. Select Shape -> Shell -> From Solid from the main menu bar to convert the solid into a shell. 2. Instance the part in the Assembly module. 3. In the Mesh module, assign Mesh Controls to Tri. Seed the part and generate a mesh. 4. Faces that cannot be meshed by ABAQUS/CAE are an indication of small faces or edges. If ABAQUS/CAE displays the message <code>Could not mesh face probably due to too coarse a mesh size</code>, reduce the seed size and generate the mesh again. If you cannot mesh a face with a reasonable seed size, you should use the manual repair tools to delete or merge the face.

Part still has invalid geometry after automated repair

Possibility	Resolution
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Limitation of repair algorithm.	<p>If only a few regions of the part are invalid, try the following:</p> <ul style="list-style-type: none"> • Select Part -> Repair Geometry and select Automated repair from the Geometry Repair Tools dialog box that appears. • Click Yes when ABAQUS/CAE asks; Do you want the invalid geometry deleted now? • Try to rebuild the missing regions using the shape feature tools and the manual repair tools.
	<p>If many regions are invalid, the precision of the geometry is probably too low. When you import the part, ABAQUS/CAE displays the IGES Options dialog box; the Minimum Resolution is displayed in the Header Information.</p> <p>To increase the precision and make the part valid, you can try clicking the IGES Options button and reducing the Scale Factor to a number less than one. The Minimum Resolution will indicate the effect of changing the scale factor. Changing the scale factor will modify the dimensions of your part, and you must adjust all other engineering quantities correspondingly.</p>

Some faces of the part are visibly incorrect

Possibility	Resolution
This is a limitation of the automated repair algorithm.	<p>Use the manual repair tools to replace faces of the part that are visibly incorrect:</p> <ul style="list-style-type: none"> • From the Geometry Repair Tools, select Remove face to remove the unwanted face or faces. • From the Geometry Repair Tools, select Create face to recreate each face. <p>Select some of the edges of the loop that define the face. If branching exists in any of the edges, select all the edges that define the face.</p>

Some faces of the part are very small

Possibility	Resolution
The part contains a small face.	Use the manual repair tools to merge the small face: <ol style="list-style-type: none"> 1. From the Geometry Repair Tools, select Remove face to remove the unwanted face. 2. From the Geometry Repair Tools, select Automated repair with the Convert to analytical representation and Stitch edges options toggled on.

IGES

- [What is IGES?](#)
- [What are the IGES options in ABAQUS/CAE?](#)
- [Changing the scale factor](#)
- [Scanning the IGES file?](#)
- [What is in the IGES log file?](#)
- [My IGES import failed. What should I do now?](#)

What is IGES?

The Initial Graphics Exchange Specification (IGES) is a neutral data format designed for graphics exchange between computer-aided design (CAD) systems. IGES was originally developed in 1980 as an ANSI standard, and it remains the most widely supported standard for storing graphics information. The original standard was designed to exchange two-dimensional drafting data, but the standard has evolved to include three-dimensional data using trimmed surface representation. The IGES standard also includes B-rep data; a B-rep in an IGES file is called a Manifold Solid B-rep Object (MSBO).

In ABAQUS/CAE you can import IGES-format parts, and you can export parts in IGES format. In addition, you can import and export a sketch from an IGES file. For a list of the IGES entities supported by ABAQUS/CAE, see "IGES entities recognized by ABAQUS/CAE when importing a part or a sketch," Section 14.7.7 in the online ABAQUS/CAE User's Manual.

What are the IGES options in ABAQUS/CAE?

The **IGES Options** tabbed page in the **Create Part from IGES File** dialog box can help you obtain a valid model for use in ABAQUS/CAE.

The **IGES Options** tabbed page allows you to choose the following:

- [Trim Curve Preference](#)
- [MSBO](#)

Trim Curve Preference

Trim curve preference controls how ABAQUS/CAE converts the surface and the trim curves into an internal representation of the part. You can choose from the following:

As per IGES file

This is the default option. When this option is selected, ABAQUS/CAE uses either of the following options to decide how the trim curve is defined. Information in the IGES file determines which of the two options is used.

Always use parametric data

This option computes the trim curve parametrically using the surface on which the curve is lying. Each of the data points on the trim curve is located by a surface parameter (u, v) . ABAQUS/CAE evaluates the surface corresponding to the data point, and then generates three-dimensional coordinates for the point.

If the underlying surface has too many sharp deflections that cannot be accurately defined parametrically, the trim curve may not lie on the surface when ABAQUS/CAE tries to reconstruct the part. This produces a trimming error and may result in gaps between edges.

Always use 3D data

This option computes the trim curve from the three-dimensional coordinates in space. Each of the data points has its own three-dimensional geometrical point; as a result the trim curve must be re-evaluated each time the surface is moved. The trim curve can move only along the surface. The **Always use 3D data** option should allow trim curves to stay with their underlying surface; however, this is not guaranteed. The **Always use 3D data** option consumes more memory than the other options; in addition, the import will take longer to complete.

MSBO

A Manifold Solid B-rep Object (MSBO, entity type 186) is an IGES term for a B-rep solid. Like all B-rep solids, the MSBO entity indicates the overall topology of a solid entity by referencing all the trimmed surfaces that define the solid.

For an IGES file to contain an MSBO entity, the CAD package that created the file must include the MSBO in the export procedure. I-DEAS and CATIA allow for the MSBO entity; SolidWorks does not.

- If you know that the IGES file does not contain an MSBO entity, you should not select the **MSBO** option from the **IGES Options** tabbed page.
- If the IGES file contains an MSBO entity, you should select the **MSBO** option and capitalize on the additional surface connectivity information provided in the entity definition.
- If the IGES file contains an MSBO entity and you do not select the **MSBO** option, the import will fail and ABAQUS/CAE will display the following message:

"This IGES file contains an entity of type 186 which sometimes cannot be read"

Scanning the IGES file

When you import an IGES-format file, ABAQUS/CAE scans the contents of the file before displaying the **Create Part from IGES File** dialog box. You can then use buttons on the **IGES Options** tabbed page to view the following:

IGES Header

The IGES header information includes details about the application that wrote the IGES file. It also includes information about the author of the file and the date when the file was written, along with the scale, resolution, and units.

Entity List

An entity can be a geometric entity, such as a point, an arc, or a line. Alternatively, an entity can be separate from the geometry, such as a comment. IGES allocates a number to each entity; for example, a circular arc is entity number 100. The IGES entity list displays a list of each type of entity found in the file. The list includes the IGES entity number, a description of the entity, and the number found in the file.

Changing the scale factor

ABAQUS/CAE uses ACIS to represent a geometric entity. ACIS has a resolution of 1.E-6; as a result if two vertices are considered coincident, they must be less than 1.E-6 units away from each other.

If you import a part from an IGES file and there is a gap slightly larger than 1.E-6 between two vertices, the model might be imprecise. However, if you reduce the scale factor before you import the part, the vertices may become coincident and the part will be precise. For example, a scale factor of 0.1 may be sufficient to make an imported part precise. This approach should be used with caution; ABAQUS/CAE

may deform the part if you specify a very small scale factor. Select the **Scale** tabbed page from the **Create Part from IGES File** dialog box to change the scale factor.

What is in the IGES log file?

When you import a part from an IGES log file, ABAQUS/CAE creates a file called `abaqus_read_iges.log` in the directory from which you started the session. The IGES log file contains information about the entities that were translated along with any problems that were encountered.

Click [here](#) to view the log file from a successful import.

Click [here](#) to view the log file from an unsuccessful import.

Import displays: "Failed to read file"

Possibility	Resolution
File does not contain any supported entities.	Look at the abaqus_read_iges.log file generated by ABAQUS/CAE. This file will be in the directory from which you started ABAQUS/CAE. If there are supported entities in the IGES file that ABAQUS/CAE attempted to read, the entities are listed at the bottom of the file under the heading "Conversion Summaries." If the file does not contain any supported entities, you must return to the CAD system and export the part in a different format or with different options.
A transcription error in the IGES file has caused ABAQUS/CAE to misread the file.	At the end of the file abaqus_read_iges.log there is a section under the heading Problem IGES Entities that lists the entities that ABAQUS/CAE could not read because of incorrect syntax. You must return to the CAD system and export the part in a different format or with different options.

Import displays:

"This IGES file contains an entity of type 186 which sometimes cannot be read"

A manifold body is a bounded, closed, and finite volume in three-dimensional Euclidean space. With Version 5.0, IGES introduced the concept of a manifold body

in the form of a Manifold Solid B-rep Object or MSBO (IGES entity type 186). An MSBO represents a solid that is made up of a set of connected trimmed and untrimmed surfaces, edges, and vertices. Additional information in the MSBO definition indicates how the surfaces are connected; as a result, an MSBO can reduce the amount of stitching that ABAQUS/CAE has to perform to reproduce the original solid.

Possibility	Resolution
File contains an MSBO entity.	Turn on the MSBO option in the IGES Options dialog box in ABAQUS/CAE.
With the MSBO option turned on, the IGES file still cannot be read.	The MSBO entity that the exporting system has written to the IGES file still cannot be recognized by ACIS. Return to the CAD system and regenerate the IGES file without using the MSBO option.

"Import displays: "File does not contain entities from which to build a part"

Possibility	Resolution
There are no IGES solid bodies in the file. Only IGES files containing entity types 144 and 186 can be guaranteed to build a solid ACIS entity in ABAQUS/CAE.	Import the file as a sketch. Or return to the CAD system and export the part using different IGES options. For more information, see What are the IGES Options in ABAQUS/CAE?

There are unwanted visible gaps between some neighboring faces

Possibility	Resolution
Incorrect trimming during import.	<p>Try a different Trim Curve Preference in the IGES Options dialog box. If you select Always use 3D data, ABAQUS/CAE should allow the surface boundary curves to stay within their surface boundaries if the IGES file contains such data. However, this option will consume more time and memory than the As per IGES file option.</p> <p>Do not select Always use 3D data for large models, because the</p>

	import process consumes large amounts of memory. In some cases memory problems will cause some entities not to be read and the import will fail before all the IGES entities have been read.
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CAD System Export Options

The following list describes the recommended formats and options that you should use when exporting a part from a CAD system. Direct translators are available for most of these CAD products that will allow you to import the geometry directly into ABAQUS/CAE. ABAQUS recommends that you use these translators rather than trying to export to a standard format, such as IGES or STEP.

CATIA Solutions

I-DEAS

HyperMesh

Pro/ENGINEER

SolidWorks

Unigraphics

AutoCAD

Other CAD products

In some cases your imported solid part will still be unusable by ABAQUS/CAE even after you use the automatic and manual repair tools. If that is the case, your only option is to return to the original CAD system that generated the exported part and do the following:

- Simplify the part.
- Try different export options.

CATIA

You can import CATIA geometry into ABAQUS/CAE using CATIA V5 and CATIA V4.

Exporting from CATIA V5

You can use the Elysium CATIA V5 to ABAQUS/CAE geometry translator to transfer geometry from CATIA V5 to ABAQUS/CAE. The translator consists of a CATIA V5 plug-in that generates an Elysium Neutral File, and an ABAQUS/CAE reader that can import the Elysium Neutral File. The plug-in is available for CATIA V5 Release 9-11.

Exporting from CATIA V4

You can use the ABAQUS direct translators to import a file saved in CATIA V4 directly into ABAQUS/CAE. If the CATIA V4 direct translator is not available at your site, contact your local ABAQUS sales office to request a copy. Alternatively, you can export the geometry from CATIA in IGES format.

Exporting from CATIA V4 to a native CATIA file

You can import a CATIA V4 geometry file directly into ABAQUS/CAE using the direct translators.

Exporting from CATIA V4 to an IGES-format file

The default in CATIA V4 is to export the boundary curves as rational B-splines (IGES entity 126). ACIS sometimes has difficulty handling a large number of these B-splines. Therefore, it is recommended that you export boundary curves as parametric splines (IGES entity 112).

To export an IGES file from CATIA V4 with parametric splines:

1. Set the spline option in CATIA to **parametric**.
2. Export the part using the CATIA->IGES translator.

I-DEAS

You can use the ABAQUS plug-in to export geometry from I-DEAS into an Elysium Neutral File that can be imported directly into ABAQUS/CAE. If the plug-in is not available at your site, contact your local ABAQUS sales office to request a copy. Alternatively, you can export the geometry from I-DEAS into a STEP- or IGES-format file that can be imported into ABAQUS/CAE.

Exporting from I-DEAS using the ABAQUS plug-in

You can export I-DEAS geometry into an Elysium Neutral File using the ABAQUS plug-in. The Elysium Neutral File can then be imported directly into ABAQUS/CAE. The plug-in is available for I-DEAS Master Series 9 and 10 NX.

Exporting from I-DEAS to a STEP- or IGES-format file

You should try to export a STEP-format file from I-DEAS before you try to export an IGES-format file.

STEP format

You can use the default settings in I-DEAS when you export to a STEP-format

file.

IGES format

When you export an IGES file from I-DEAS, it is important that the IGES file be flavored properly. In most cases ABAQUS/CAE will generate error messages when you attempt to import an IGES file written in an incorrect flavor. Click [here](#) to download the flavor file. This file specifies the settings recommended for optimal import and healing in ABAQUS/CAE.

To export an IGES file from I-DEAS Master Series 8:

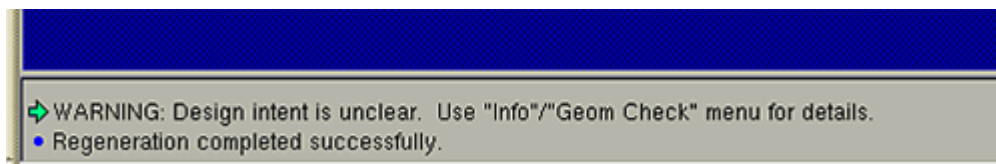
1. Place the [ABAQUS flavor file](#) in the directory in which you are running I-DEAS.
2. Start I-DEAS.
3. From the I-DEAS main menu, select **File->Export**.
4. In the **Export Selections** dialog box, select **IGES** and click **OK**.
5. Select the part that you want to export.
6. Under the **Apply Flavor** label, toggle on **Local** and select **HKS** as the flavor.
7. Click **Start Export**. An IGES file is generated in the ABAQUS flavor.

HyperMesh

Use the 'JAMA planes' option to export geometry in IGES format from HyperMesh for import into ABAQUS/CAE.

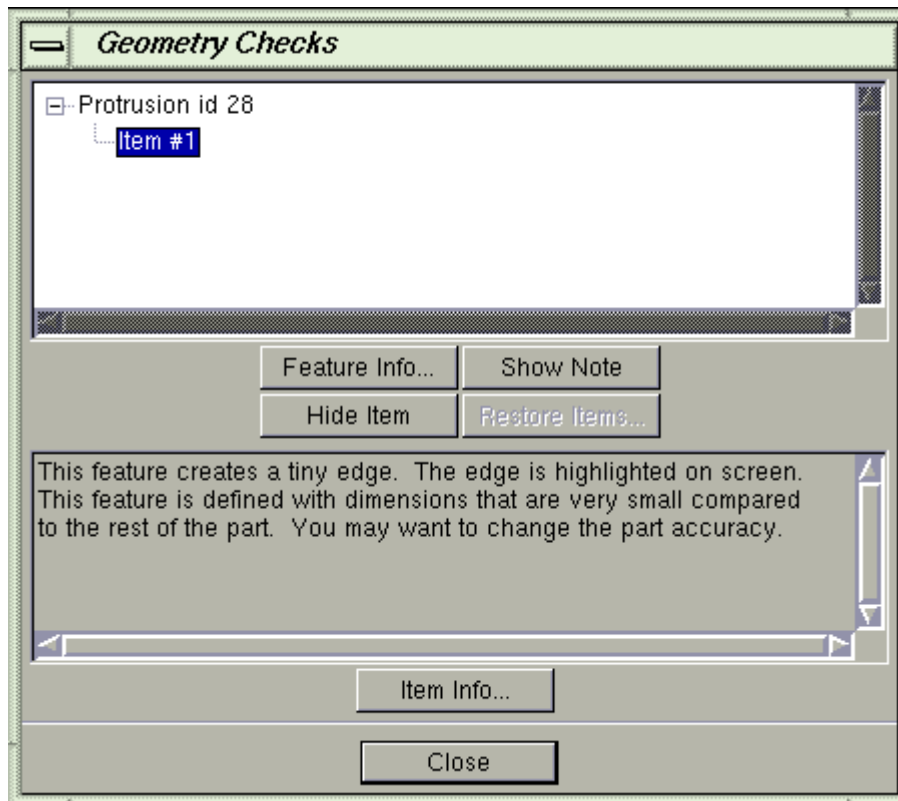
Creating a Valid Pro/Engineer part

If you see the message shown in the following figure after you regenerate a part in Pro/ENGINEER, you may experience problems importing the part into ABAQUS/CAE.



The part may have regenerated successfully; however, you should select **Info->Geom Check** from the Pro/Engineer main menu bar to check for problems that might cause import into ABAQUS/CAE to fail.

After you select **Info->Geom Check**, Pro/ENGINEER displays the **Geometry Checks** dialog box, as shown in the following figure:



The geometry checking process in Pro/ENGINEER looks for the following:

- Features that are very small compared to the rest of the part.
- Surfaces being merged that do not match exactly.
- Radii that are too large to round an edge.
- Features that refer to an edge that is not present in the geometry.
- Features that refer to an edge that was already rounded by another feature.
- Features that refer to an edge that was not created during the last regeneration because the part was modified.
- Features that created an edge and have been deleted.
- Sections where the beginning or ending position of the section is very close to, but not touching, a vertex.
- Features that create a tiny edge. The feature was sketched on a drafted plane, which might be responsible for the problem. A previously existing edge was almost completely trimmed away, creating a tiny edge. The system could not construct the intersection of the part and the feature.
- Features that contain self-intersecting surfaces. A surface has overlapping geometry.
- Features that cut tiny pieces from an existing edge or edges. A feature can also leave tiny pieces of an existing edge.
- Surfaces that contain a singularity.

- Changes that lead to misalignment near points.

Exporting from Pro/Engineer

You can use the ABAQUS plug-in to export geometry from Pro/ENGINEER into an Elysium Neutral File that can be imported directly into ABAQUS/CAE. If the plug-in is not available at your site, contact your local ABAQUS sales office to request a copy. Alternatively, you can export the geometry from Pro/ENGINEER into a STEP- or IGES-format file that can be imported into ABAQUS/CAE.

Exporting from Pro/ENGINEER using the ABAQUS plug-in

You can export Pro/ENGINEER geometry into an Elysium Neutral File using the ABAQUS plug-in. The Elysium Neutral File can then be imported directly into ABAQUS/CAE. The plug-in is available for Pro/ENGINEER Versions 2001 and Wildfire.

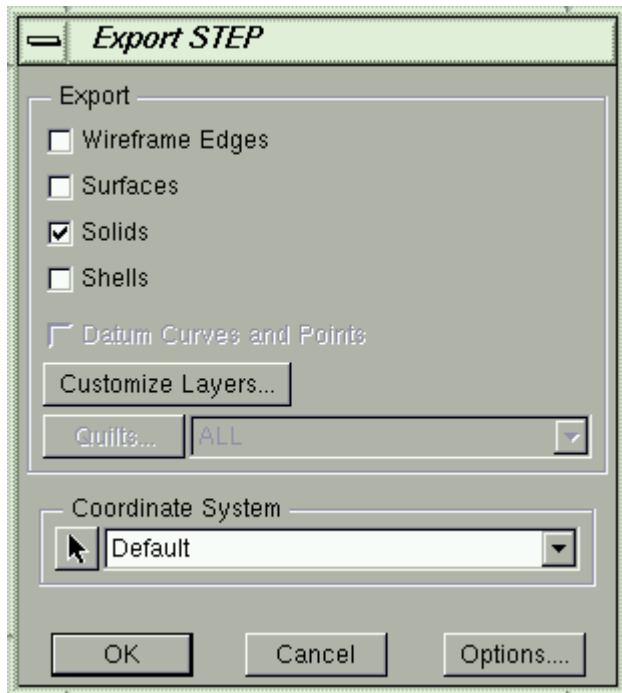
Exporting from Pro/ENGINEER to a STEP- or IGES-format file

You should try to export a STEP-format file from Pro/ENGINEER before you try to export an IGES-format file.

STEP format

When exporting a STEP file from Pro/ENGINEER, it is important that the "solids" option is toggled on in the export options. This setting is required even for surface geometry because when the "solids" option is selected, the topological information about the geometry is exported to the STEP file. Having this topological information present in the STEP file will substantially increase the chances of success when importing the part into ABAQUS/CAE.

The **Export STEP** dialog box containing the optimal settings in Pro/Engineer is shown below.



IGES format

The following list describes some of the problems that you might encounter when you import a Pro/ENGINEER IGES-format file into ABAQUS/CAE along with the recommended settings to avoid such problems. IGES files are described in detail in "Understanding the contents of an IGES file," Section 14.4, in the ABAQUS/CAE User's Manual.

Symptom: Poor handling of trimmed and bounded surfaces

This message indicates that ABAQUS/CAE was unable to create a B-rep solid using the trimmed curves. Use the default IGES settings in Pro/ENGINEER, and do one of the following:

- Include the following options in the IGES-specific configuration file

iges_config.pro:

- iges_out_trim_deviation 0.000001
- iges_out_trim_xyz yes
- intf3d_out_extend_surface no

Change the trim curve preference in ABAQUS/CAE to **Always use 3D data**.

- Include the following options in the IGES-specific configuration file

iges_config.pro:

- iges_out_trim_xyz no
- intf3d_out_extend_surface no

Change the trim curve preference in ABAQUS/CAE to **As per IGES file**.

Symptom: Entire assembly is read in as one part

By default, Pro/ENGINEER exports an assembly to a single file containing a single part. You can change this behavior by setting a configuration option that instructs Pro/ENGINEER to output an assembly to IGES as multiple files containing geometry information for each part. This option also instructs Pro/ENGINEER to export any assembly features. If you set this option, ABAQUS/CAE can import each part individually from the separate IGES files.

Include the following option in the IGES-specific configuration file `iges_config.pro`:

- `iges_out_assembly_default_mode all_parts`

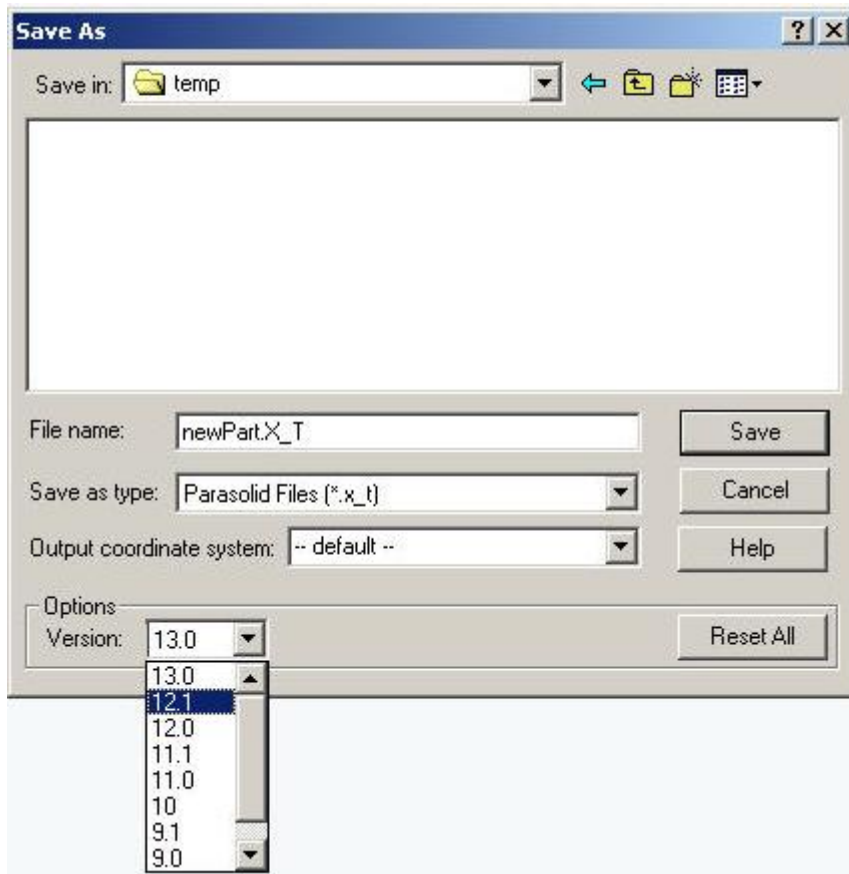
SolidWorks

You can use the ABAQUS direct translators to import SolidWorks native Parasolid geometry directly into ABAQUS/CAE. If the Parasolid direct translator is not available at your site, contact your local ABAQUS sales office to request a copy. Alternatively, you can export the geometry from SolidWorks into an ACIS-format (.sat) file that can be imported into ABAQUS/CAE.

Exporting from SolidWorks using the native Parasolid format

You can import a Parasolid geometry file directly into ABAQUS/CAE using the direct translators. ABAQUS/CAE supports Parasolid Version 7 through 14.

When you save the SolidWorks geometry as a Parasolid file, you must select a valid version of Parasolid from the **Export Options** dialog box, as shown in the following figure:



Exporting from SolidWorks 2001 Plus to an ACIS-format file

When you export an ACIS (. sat) file from SolidWorks 2001 Plus, you should accept the default ACIS Version option of **7.0** from the **Export Options** dialog box, as shown in the following figure:

Unigraphics

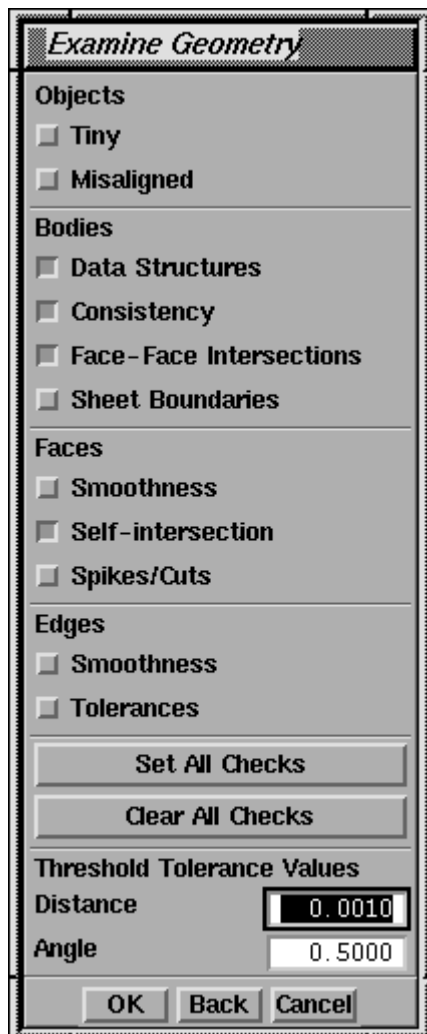
You must ensure that the Unigraphics part that you are trying to export to ABAQUS/CAE is a valid part.

Creating a valid Unigraphics part

Before you export a part from Unigraphics, examine the part's geometry to make sure that there are no errors corrupting the part. Files generated from corrupt parts are the most common cause of problems when importing a part from Unigraphics into ABAQUS/CAE.

To examine a part's geometry in Unigraphics:

1. From the main menu bar, select **Analysis -> Examine geometry**.
The **Examine geometry** dialog box appears.
2. Toggle on the options shown below:



3. Click OK.

Unigraphics will run a check on the part(s) and will give diagnostics of any errors it finds. You must fix any errors in the geometry before exporting a part to IGES. The [ProSTEP](#) website contains more detailed information on checking geometry in Unigraphics.

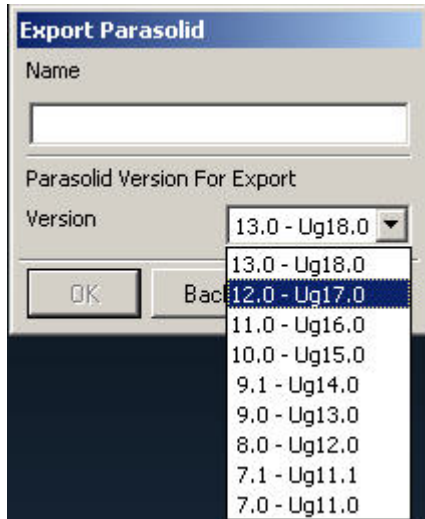
Exporting from Unigraphics

You can use the ABAQUS direct translators to import Unigraphics native Parasolid geometry directly into ABAQUS/CAE. If the Parasolid direct translator is not available at your site, contact your local ABAQUS sales office to request a copy. Alternatively, you can export the geometry from Unigraphics into a Step- or IGES-format file that can be imported into ABAQUS/CAE.

Exporting from Unigraphics using the native Parasolid format

You can import a Parasolid geometry file directly into ABAQUS/CAE using the direct translators. ABAQUS/CAE supports Parasolid Version 7 through 14.

When you save the Unigraphics geometry as a Parasolid file, you must select a valid version of Parasolid, as shown in the following figure:



Exporting from Unigraphics to a STEP- or IGES-format file

You should try to export a STEP 203/214-format file from Unigraphics before you try to export an IGES-format file.

STEP format

You can use the default settings in Unigraphics when you export to a STEP-format file.

IGES format

When you write IGES files with Unigraphics, ABAQUS recommends certain export settings. These settings will maximize the success rate when importing into ABAQUS/CAE. These settings differ significantly from the default Unigraphics export settings. Click [here](#) to download a file containing the recommended settings.

To export an IGES file from Unigraphics:

1. Download the recommended [settings file](#).
2. From the File menu, select **File -> Export -> IGES**. Unigraphics displays the **Export IGES** dialog box.
3. In the dialog box, click the **Class select** button.
4. In the viewport, click the part that you want to export, and click **OK**.

5. Click the **Choose settings** button.
6. Select the settings file that you downloaded in Step 1.
7. Click **OK**.

The Unigraphics->IGES translator will be initialized. The time taken to translate the part depends on the size and/or the complexity of the part being translated.

AutoCAD

AutoCAD, like ABAQUS/CAE, uses ACIS as the internal geometry format. Because AutoCAD uses ACIS, the ACIS (.sat) file format is the most efficient and reliable means of transferring parts between AutoCAD and ABAQUS/CAE.

To export an ACIS (.sat) file from AutoCAD:

1. From the **File** menu in AutoCAD, select **Export**. AutoCAD displays the **Export Data** dialog box.
2. In the dialog box, enter a file name and select **ACIS (*.sat)** from the **Save as type** list.
3. Click **Save**.
4. Select the objects to export, and press **Enter**.

Importing geometry from other CAD products into ABAQUS/CAE

Many CAD products--such as Solid Edge, FEMAP, and MSC.Patran--can generate Parasolid-format parts that can be imported into ABAQUS/CAE using the ABAQUS/CAE direct translator. ABAQUS/CAE supports Parasolid Version 7 through 14.

If the direct translator is not available, you can try to export the part using another format. The order of preference is:

1. ACIS
2. STEP
3. IGES
4. VDA-FS

Frequently asked import and repair questions

Import

Q: The IGES Entity Filter list shows unsupported or unknown entities. Should I try to import/repair the part in ABAQUS/CAE or try to regenerate the part in the CAD package without creating the unsupported entities?

A: The **IGES Options** dialog box includes an **Entity Filter** that lists all the IGES entities found in the file. If the **Entity Filter** includes nongeometric entities that are labeled as **Unsupported/Unknown**, you cannot be sure how much of the part has been imported. You should return to the CAD system and export the part again with different options; however, if only a few entities are missing, you may be able to repair the part using the automated and manual repair tools.

Q: When should I try using the Trim Curve options?

A: If ABAQUS/CAE displays silhouette lines or shadowy surfaces in shaded render mode after you import a part, it is an indication that the trim curves were not imported correctly. In general, trying the different **Trim Curve Preferences** will allow ABAQUS/CAE to create the best fit between the curve and its surface data.

Most of the time, the IGES file will have a trim curve preference in the IGES Entity 142 record. The trim curve preference is the final item at the end of the record; you can determine the preference as follows:

- 0 Unspecified
- 1 Two-dimensional trim curve
- 2 Three-dimensional trim curve
- 3 Either curve is equally preferred

Q: When should I use the MSBO option?

A: If a part has been exported to an IGES file with the MSBO entity, you must select the **MSBO** option in ABAQUS/CAE to read the additional B-rep information contained in the file. If you do not know how the IGES file was created, ABAQUS/CAE will display a message if the **MSBO** option is turned off and the file contains an MSBO entity.

Repair

Q: Does it make any difference if I repair the part during or after import?

A: No.

Q: When should I try to use only the "Convert to analytical representation" repair option?

A: You should never use only the **Convert to analytical representation** repair option. You should always select the **Convert to analytical representation** option in conjunction with the **Stitch edges** option.

Q: Can I select only the "Convert to precise representation" repair option?

A: No. ABAQUS/CAE selects the **Convert to analytical representation** option and the **Stitch edges** option when you select the **Convert to precise representation** option.

Q: I tried stitching the part, but it is still imprecise. What should I try next?

A: First use the **Query** toolset to highlight the imprecise entities. By highlighting the imprecise entities, it is easier to judge what action to take. You have the following options:

- If there are many imprecise entities, you should return to the CAD system and try to export the file with different options.
- If the part is small and does not contain many features, use the **Convert to precise representation** option to make the part precise. Making the part precise is time consuming and is not always successful. In addition, making the part precise may distort the geometry.
- Repair the part using the manual repair tools and stitch the part again.
- If there are a few imprecise entities, you may be able to continue working with the part. If you mesh the part, ABAQUS/CAE will indicate the problematic areas.

Q: Does reducing the feature list help in repairing the part?

A: If ABAQUS/CAE cannot repair one of the features correctly, reducing the feature list may help. In general, the part described in the imported file is treated as a single feature. As a result, reducing the feature list may delete the part you are trying to import.

Q: The repair I tried made things worse. What do I do now?

A: ABAQUS/CAE stores repair operations as features. As a result, you can undo a repair by deleting the feature corresponding to the repair.