



**GIBBSCAM 2026** CAM for  
Production Machining

Version 2026 : September 2025

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## Data Exchange



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# Introduction

This [Data Exchange](#) guide describes all forms of import and export that are available to you. Although many options are available, your package is limited to the options purchased.

**Note:** The capabilities and user interface described in this and other guides apply to GibbsCAM Industrial Edition with all product options licensed and active. GibbsCAM Viewer and GibbsCAM Student Edition provide a subset of the full functionality.

Data Exchange can be used in several ways:

- Directly read geometry, surface-model, and solid-model formats generated by other CAD packages.
- Use CAD Add-Ins that provide for the direct transfer of parts from their native program.
- Import and interpret attributes (such as colors or Hole features) generated by another package.
- Add files to a model by importing.
- Send models to other formats by exporting.

Several methods for exchanging files in many formats are available. Models can be modified in a previous version of GibbsCAM, on another computer, or even on another program.

This guide assumes a level of proficiency with geometry creation, coordinate systems, and basic machining. Before reviewing this document, you should already be familiar with the basics of GibbsCAM as outlined in [Getting Started](#), [Geometry Creation](#), and [Mill](#) or [Turning](#). Additionally, if you will be using solids, it will be very helpful if you have a basic understanding of [Advanced CS](#) and any of the solids packages ([Solids Import](#), [2.5D Solids](#), or [SolidSurfacer](#)).

Open/Import Formats	Export Formats	CAD Add-Ins
<ul style="list-style-type: none"> <li>• ACIS / SAT (filetypes *.sat, *.sab, *.asat, and *.asab)</li> <li>• AutoDesk RealDWG (filetypes *.dwg and *.dxf*)</li> <li>• AutoDesk Inventor (Spatial) (filetypes *.ipt and *.iam)</li> <li>• CATIA v4 (filetypes *.DLV, *.model and *.exp)</li> <li>• CATIA V5/V6 (filetypes *.CATPart and *.CATProduct)</li> <li>• PTC Creo Parametric Part (filetypes *.PRT and *.PRT*) and Assembly (filetypes *.ASM and *.ASM*)</li> <li>• DWG</li> <li>• DXF (filetype *.dxf)</li> <li>• IGES (filetype *.igs and *.iges)</li> <li>• KeyCreator (filetype *.CKD)</li> <li>• NX (filetype *.prt); also called UGS/NX</li> </ul>	<ul style="list-style-type: none"> <li>• ACIS / SAT v6 through v29 (filetype *.sat)</li> <li>• PTC Creo Parametric ATB (filetype *.neu)</li> <li>• DXF (filetype *.dxf)</li> <li>• IGES (filetype *.igs)</li> <li>• Parasolid v8 through v33 (filetype *.x_t)</li> <li>• point list (filetype *.txt)</li> <li>• STL (filetype *.stl)</li> <li>• STL binary (filetype *.stl)</li> <li>• STEP AP203, AP214, and</li> </ul>	<ul style="list-style-type: none"> <li>• Autodesk Inventor 2010 and newer (64-bit)</li> <li>• Solid Edge 100 (ST) and newer</li> <li>• SOLIDWORKS 2011 and newer</li> </ul>

Open/Import Formats	Export Formats	CAD Add-Ins
<ul style="list-style-type: none"> <li>Parasolid (filetypes <b>*.x_t</b> and <b>*.xmt</b>), Parasolid binary (filetype <b>*.x_b</b>), and Parasolid Partition (filetype <b>*.p_b</b>)</li> <li>point list (filetype <b>*.txt</b>)</li> <li>Rhinoceros Open Nurbs (filetype <b>*.3DM</b>)</li> <li>Solid Edge (filetype <b>*.par</b>) and Solid Edge Assembly (filetype <b>*.ASM</b>)</li> <li>SolidWorks (filetype <b>*.SLDPRT</b>) and SolidWorks Assembly (filetype <b>*.SLDASM</b>)</li> <li>STEP AP203/AP214/AP242 (filetypes <b>*.STEP</b> and <b>*.STP</b>)</li> <li>STL (filetype <b>*.stl</b>)</li> <li>VDA-FS (filetype <b>*.vda</b>)</li> </ul>	AP242 (filetype <b>*.STEP</b> )	

Name changes:

- Pro/ENGINEER: *see PTC Creo Parametric*
- Creo Elements/Pro: *see PTC Creo Parametric*
- Siemens PLM: *see NX*
- Unigraphics / UGS: *see NX*
- Wildfire: *see PTC Creo Parametric*

Other notes:

- Add-ins for exchanging data between GibbsCAM and Cimatron are now supplied by Cimatron.

Import	version	Export	version	Add-In	version
ACIS	R1 → 2023 1.0	ACIS	24		
DXF/DWG	2.5 → 2023	DXF			
Pro/E or Creo	16 → Creo 10.0	IGES		Autodesk Inventor	2010 and newer
CATIA V4 CATIA V5/V6	4.1.9 → 4.2.4 V5R8 → V5-6 R2024	Parasolid	9.0 → 35.0.149	Solid Edge	2021 and newer
Inventor	V11 → 2025	STEP		SolidWorks	100 (ST) and newer
NX	11 → NX 2306	3D PDF			
Parasolid	35	PTC Creo Parametric			
SolidWorks	98 → 2024*	STL			

Import	version	Export	version	Add-In	version
Solid Edge	V18 → SE 2024				
STEP	AP203, AP214, AP242				
IGES	up to 5.3				
Rhino	up to version 7				
STL	N/A				
VDA-FS	1.0 → 2.0				
KeyCreator	2022				

## Notes:

- SolidWorks 98 through SolidWorks 2024 are supported through **Spatial** (a unit of Dassault Systèmes, and developer of the 3D ACIS kernel).

## Name changes:

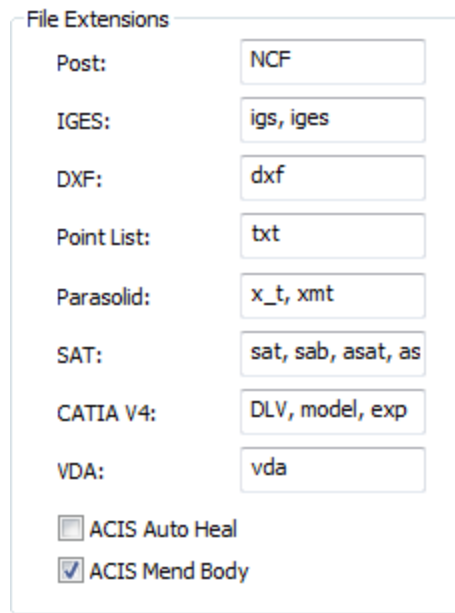
- Pro/ENGINEER: *see PTC Creo Parametric*
- Creo Elements/Pro: *see PTC Creo Parametric*
- Siemens PLM: *see NX*
- Unigraphics/UGS: *see NX*
- Wildfire: *see PTC Creo Parametric*
- Add-ins for exchanging data between GibbsCAM and Cimatron are now supplied by Cimatron.

# Interface

This chapter describes the basic interface items that are specific to importing and exporting data. The details of the interface for importing and exporting are covered below.

- [File Extensions](#)
- [“Files Without Extensions” on page 9](#)
- [“File Import and File Export” on page 9](#)
- [“CAD Attribute Import” on page 9](#)
- [“Option Dialogs” on page 10](#)

## File Extensions



File Extensions

Post:	NCF
IGES:	igs, iges
DXF:	dxf
Point List:	txt
Parasolid:	x_t, xmt
SAT:	sat, sab, asat, as
CATIA V4:	DLV, model, exp
VDA:	vda

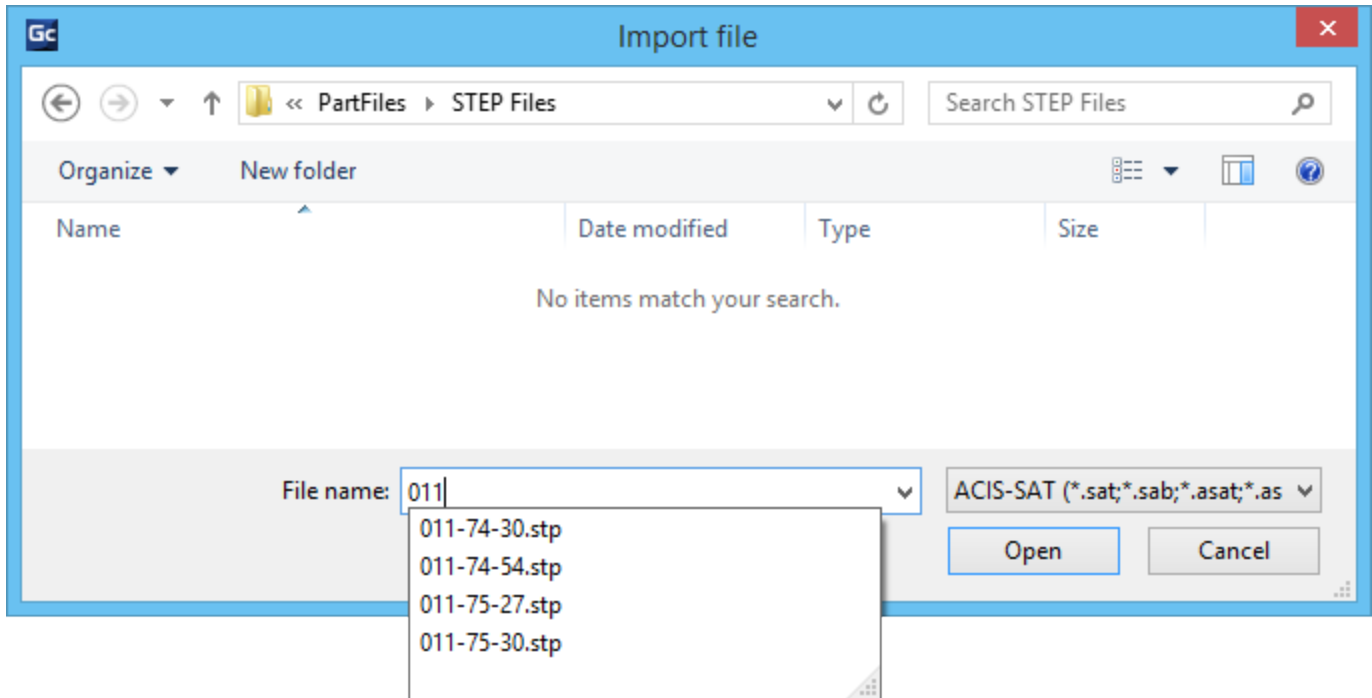
☐ ACIS Auto Heal  
☒ ACIS Mend Body

Every file should have an extension type that defines what type of file it is. Native GibbsCAM files have a three letter extension vnc which stands for “Virtual Numeric Control.” Parasolid files have different extensions depending on the CAD program that was used to generate the file. For example, Parasolid files generated in SolidWorks have the extension x\_t. Parasolid files generated by Unigraphics have an extension of xmt. Because many users and applications change their extension name for various designation reasons, GibbsCAM offers a way to handle these differences.

The File > Preferences > File Extensions dialog designates the extension for exchange files. In order to open a file with a different file extension, the extension of the file to be imported must match the extension entered in this dialog. The default extensions for each type are contained in the dialog. To add a file extension, use a comma to separate the values as shown in the Parasolid or Catia fields.

The extension displayed next to the file type in both the Open part, Import, and Export dialogs will match the extension entered in the File Extensions dialog. For example, if x\_t is the only entry for the Parasolid extension, it will only recognize Parasolid files with that particular extension.

## Files Without Extensions



If a file does not have an extension, the **Open part file** and **Import** dialogs can open such files even though they do not have extensions or if they have the wrong extension specified. Simply type the first few letters of the filename, choose the file from the drop-down menu, set the file type, and click the **Open** button. The file will be opened as if it were labeled properly.

## File Import and File Export

The import and export packages (including the Exchange option) provide the capability to transfer model data from other CAD packages into GibbsCAM. The **File > Import** dialog lets you select the type of file to be imported. 2D geometric and 3D model files can be imported into an existing file. Imported geometry will be placed in a new workgroup. The new workgroups will be viewed as visible and inactive.

## CAD Attribute Import

When you import models from other systems, CAD attributes are preserved. You can map CAD attributes to GibbsCAM user attributes when you initially import the model, or at any time thereafter (using **Features > Attribute Manager**). When you import or reimport, you can modify user attribute definitions. For details, see [CAD Attribute Mapping](#).

To map CAD attributes when importing a CAD model

1. Click **File > Import** and select a CAD model to import.
2. In the Solid File Import Options dialog, select one or more solid file import options and configurations. For more information, see [Option Dialogs](#).

3. In the Define CAD Attribute Map dialog, select one or more checkboxes for the CAD attributes you want to map, and then click **OK**.

## CAD Attribute Mapping

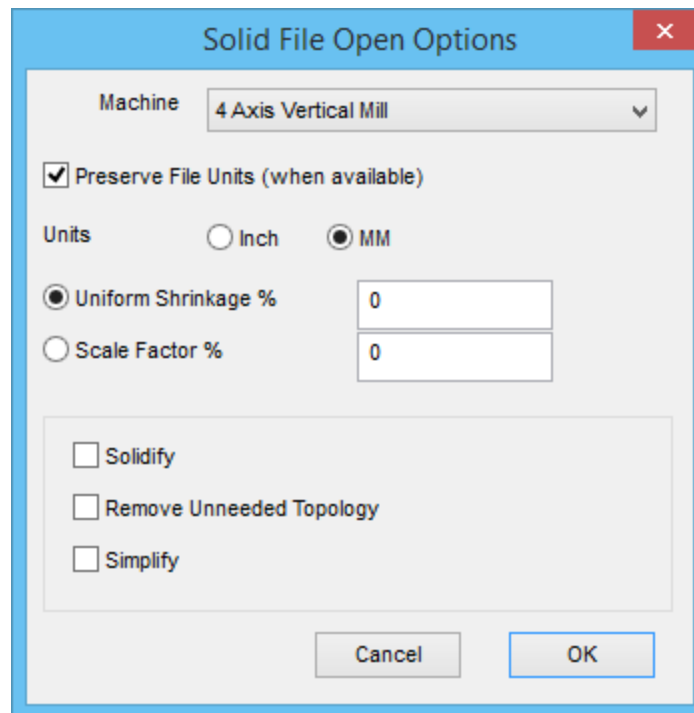
When you import CAD attributes, you normally retain the system attribute name and attribute type without changes. After a system attribute is mapped to a particular name and type, re-mapping affects the attribute's values only, not its name or type.

Note: To re-map to a different attribute name or attribute type, you must first delete the existing user attribute.

System Attribute Type	New Type	Description
Integer	Real or Text	Each integer value is retained and translated. For example, the integer 1 becomes real 1.0000 or text string "1.0000".
Integer	Color	The first 64 unique integer values are mapped to 64 different colors; the next 64 unique integer values are mapped to the same set of 64 colors; and so forth.
Integer	Feature	Each integer value is mapped to a new feature ID, but feature colors are reused in cycles of 64.
Real or Text	Integer	The real or text values become undefined.
Color	Integer	White is mapped to 1 and other colors are mapped to 0.
Color	Real (or Text)	White is mapped to 1.0000 (or "1.0000") and other colors are mapped to positive real values less than 1.0000.

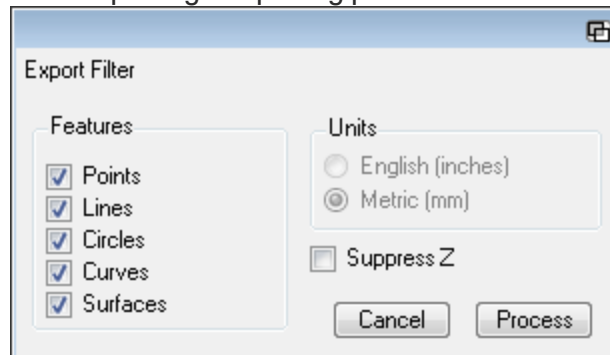
## Option Dialogs

When a surface or solid file is opened or imported, a dialog offers options for handling bodies.



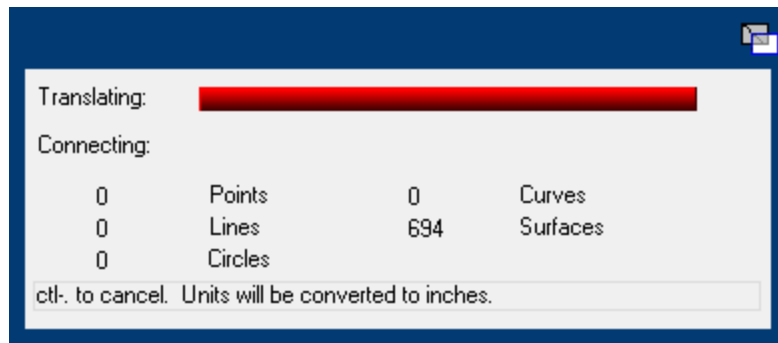
## Filter and Info dialogs

Depending on the file types you open or import, additional dialogs provide you with more information and control when importing or opening parts.



## Translation

When you are finished entering information in the Import and Export dialogs, click on the Process button. The translation dialog will appear on the screen to indicate that the file is being translated. The progress for each type of element will be shown.



# Import

GibbsCAM import features provides a way to bring part models designed with different CAD packages into GibbsCAM. This section details all of the importing capabilities of GibbsCAM. Importing can be accomplished in two ways, by direct transfer (File > Open) or by adding a model to a pre-existing GibbsCAM file (File > Import).

## Terms

### Exchange

Any action that leads to accessing a model from another application whether opening/importing or using the associative transfer add-ins.

### Feature

A piece of geometry; a point, line, circle, curve, etc. in an imported file.

### Workplane

The 3D plane in which a specific feature was defined; similar to the use of coordinate systems in GibbsCAM.

### Level/Layer

A grouping of features. IGES files use the term “level” while DXF files use the term “layer”; similar to workgroups.

### Coordinate System

A plane in space with an origin and three axes (horizontal, vertical and depth). Every feature lies in a coordinate system. For example, a point is a location in space defined by three numbers, a horizontal coordinate, a vertical coordinate and a depth coordinate.

### CS1

In a mill machine CS1 is the coordinate system where the horizontal, vertical and depth axes correspond to the X, Y, and Z respectively. In a lathe, CS1’s horizontal, vertical and depth axes correspond to X, Z, and Y respectively.



When importing files it is often necessary to move the part origin. If there is a point at the desired origin, the easiest way to move the origin is to select Modify > Move Part Origin menu and use the interrogate a point or enter the correct origin. More information may be found in the [Common Reference](#) guide.

## File > Open

File exchange in GibbsCAM is typically accomplished using the Open dialog. If your system includes solids (Solids Import, 2.5D Solids, and SolidSurfacer), an options dialog appears when you open an imported file containing a body (in other words, a surface or a solid). Depending on the type of file you open, this dialog lets you select a machine type, override the part units (millimeters or inches), and apply shrinkage and scale factors. It also offers options so that surface parts can be automatically stitched (using the Solidify command), and bodies can be automatically simplified or have unneeded topology removed.

**Solidify**

This option will attempt to convert a surfaces model into a solid model. If this function fails you can use the Solidify function to attempt the solidify using other options.

**Remove Unneeded Topology**

This command will inspect selected bodies or faces for any unneeded elements. If two items share the same underlying surface, the body will be simplified. This is identical to the command found in the Solids menu.

**Simplify**

This command simplifies the geometry of a body wherever possible. A body will be defined by lines, circles, spheres, cubes, cylinders, and toruses instead of B-splines. This is identical to the command found in the Solids menu.

## Geometry-Based Files

The Exchange option allows geometric parts to be transferred from CAD systems to GibbsCAM using the Import command. Exchange enables the import (and export) DXF, DWG, IGES and Point List files.

## DXF

DXF is the most common geometry format in most CAD applications. DXF files can be imported into a GibbsCAM part or opened directly. DXF files are plain text files that can be modified or viewed with any text editor. It is often easier to edit some DXF problems in a text editor rather than recreating or exporting the file again.

## DXF Problems

If there is a problem with a DXF file the first step would be to verify the contents by opening it in a text editor. The file should look similar to the following:

```
0
```

```
SECTION
```

```
2
```

```
HEADER
```

```
...(DATA)...
```

```
EOF
```

Each file should have these entries and should end the file with the EOF (End of File) call. Files that appear as follows were most likely created in a UNIX environment and are all but useless. You should have the creator of the file re-export the file for Windows. This will convert line endings from the UNIX format to a DOS format.

```
0SECTION2HEADER...(DATA)...EOF
```

The majority of problems are simply a misuse of the file extension. Often DXF files are really DWG or binary DXF files. Users often attempt to open compressed files with ZIP or EXE extensions by renaming them with a DWG extension. Sometimes the file may be received with these problems. However, using the DWG filter will open renamed most DXF files.

## DWG

DWG (DraWinG) files can be imported into a GibbsCAM part or opened directly. A DWG file is an encoded version of a DXF file. Since DWG is a native format there may be issues in newer versions. For most uses, DXF is recommended.

## About DWG and DXF

DWG and DXF files were developed by AutoDesk. DWG (DraWinG) is the native file type of AutoCAD and DXF (Drawing eXchange Format) is a commonly used means for exchanging drawings between CAD systems. A DXF file is much larger than its DWG counterpart and carries with it some inconveniences. DXF files can not be synched for inter-operability, ensuring the most recent updates to the file. DXF data can also be imprecise due to the smaller number of decimal places, compared to its native DWG counterpart. Knowing the details of the creation application will make the data exchanging these formats easier.

Entity	Feature Equivalent
POINT	Point
LINE	Line
3DLINE	Line
CIRCLE	Circle
ARC	Circle
POLYLINE	Connected Lines
3DPOLYLINE	Connected Lines

## Autodesk comment on DWG

“Since the AutoCAD drawing database (.DWG file) is written in a compact format that changes significantly as new features are added to AutoCAD, we do not document its format and do not recommend that you attempt to write programs to read it directly. To assist in interchanging drawings between AutoCAD and other programs, a Drawing Interchange file format (DXF) has been defined. All implementations of AutoCAD accept this format and are able to convert it to and from their internal drawing file representation.” (Autodesk)

## IGES

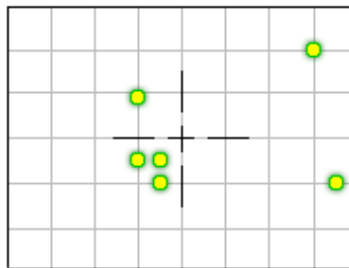
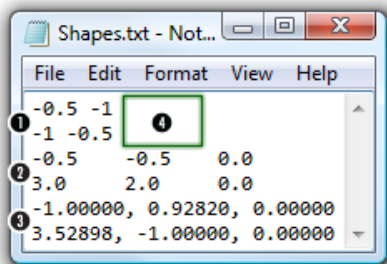
IGES (Initial Graphics Exchange Specification) is supported by many CAD systems and is published by the US Product Data Association.

The table shows the geometric IGES entities supported for importing from an IGES file. IGES files are treated the same as DXF and DWG files with the same filters as mentioned previously.

Entity #	Entity Name	Feature Equivalent
100	Circular Arc	Circle
102	Composite Curve	Connected features
104	Conic Arc	B-Spline
106	Copious Data	Connected lines
108	Plane (Bounded; Form 1)	
110	Line	Line
112	Parametric Spline Curve	B-Spline
116	Point	Point
126	Rational B-Spline Curve	B-Spline
130	Offset Curve	B-Spline

## Point List

A point list is a simple text file containing X, Y, and Z coordinates. (Note that Z coordinates are not required for GibbsCAM to understand the list.) Each point should be on a line by itself. Delimiters to separate the numbers can be tabs, spaces, or commas with spaces. When creating the point list, DO NOT create labels for X, Y, and Z. The following example point list geometry is taken from exercise #1 of [Geometry Creation](#).



1. Space separated values
2. Tab-separated values
3. Comma+space separated values
4. Z value not entered

Example of a point list file using various separators and the points it generates.

## Solid-Based Files

Solid file formats, such as Parasolid, are the most simple method for data exchange as systems sharing the same technology do not require translation as do other file types. Solid models provide a more complete and accurate part description. Solid models may be opened or imported. When importing, a model will be added to the current file.

# Parasolid

## x\_t and xmt Files

Parasolid files are native to GibbsCAM and there is no translation involved when they are brought in. This is due to the fact that GibbsCAM is built with Parasolid technology, providing for a direct reading of Parasolid information. GibbsCAM supports the flat assemblies option in Parasolid files, but not hierarchical assemblies.

## Solid Edge

If you have a copy of Solid Edge installed on your system, you can also transfer Solid Edge parts and assemblies from within Solid Edge. For more information, see [“Solid Edge Transfer Add-In” on page 33](#).

## .PAR and .ASM Files

For opening and importing purposes, Solid Edge part and assembly files are treated exactly the same as Parasolid files. If you open or import an assembly (.ASM file), all the parts (.PAR files) that it references must reside in the same folder.

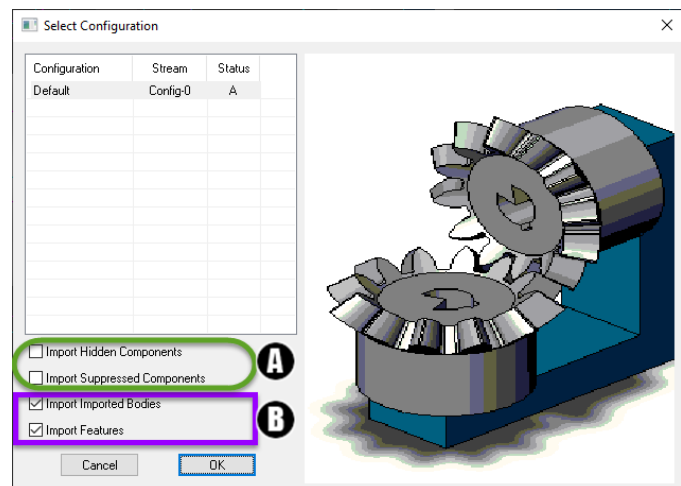
## SolidWorks

### .sldprt Files

For opening and importing purposes, SolidWorks models (v5.0 or later) are treated exactly the same as Parasolid files. If you have a copy of SolidWorks installed on your system, see [“SolidWorks Transfer Add-In” on page 33](#).

SolidWorks files can contain many revisions of a part. These revisions are referred to as “configurations”. When a SolidWorks part is exchanged, a list presents all available configurations, and a preview is available for each configuration.

Note that the File > Preferences dialog allows you to choose between translators: **Solidworks**, or **Spatial** (a unit of Dassault Systèmes, and developer of the 3D ACIS kernel). Models imported by the Spatial translator allow you to access PMI data created by SolidWorks DimXpert. If the SolidWorks translator is in effect, the **Select Configuration** dialog offers four checkboxes (A and B in the illustration). If the Spatial translator is in effect, the dialog offers two checkboxes (A in the illustration) and always imports imported bodies and features if the \*.sldprt or \*.sldasm file contains them.



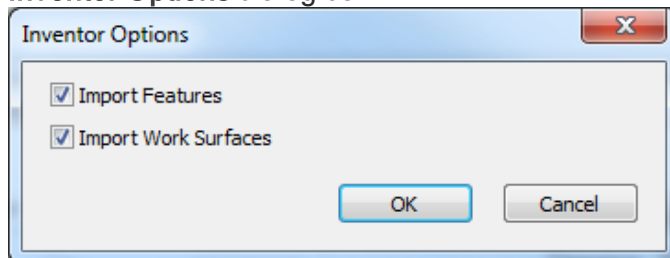
# Autodesk Inventor

## .ipt and .iam Files

Autodesk supplies a free viewer for opening \*.iam or \*.ipt files. For Autodesk Inventor 5 through Autodesk Inventor 10, this was **Design Tracking**. For later Inventor formats, it is **Inventor View**. Both are available as either a plug-in to Autodesk Inventor or as a free standalone application. Autodesk Inventor software can be downloaded from <http://www.autodesk.com/inventor>. GibbsCAM can import Autodesk Inventor solid model files only if the GibbsCAM workstation has installed the viewer plug-in or the standalone application. Also, more specifically: To transfer or import Autodesk Inventor *features* into GibbsCAM, the level of Autodesk **Inventor** or **Inventor View** must be **2014** or later. You can store Autodesk Inventor features in a way that makes them understandable to GibbsCAM.

## What Are Autodesk Inventor Features?

Autodesk Inventor defines several kinds of features: holes, extrusions, fillets, chamfers, and so forth. Features in Autodesk Inventor part files (\*.ipt) and assembly files (\*.iam) can be understood by GibbsCAM 2026, but only if they were saved while the GibbsCAM CAD Transfer Add-In for Autodesk Inventor was installed on the Autodesk workstation. Such feature data can sometimes be quite large, and it is not always needed. In GibbsCAM, you can optionally set a switch that suppresses the importation of such feature data, using the **Inventor Options** dialog box:



To open this dialog box: Plug-Ins > (Misc) > Autodesk Inventor Options

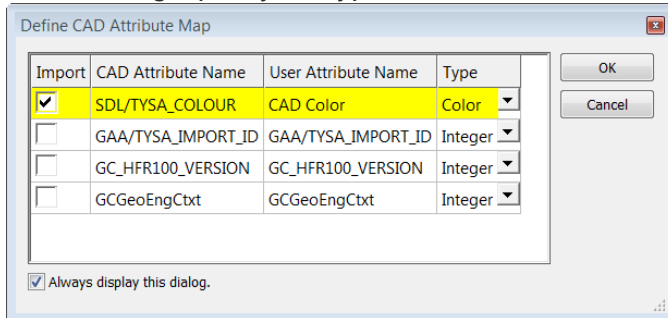
## Importing Autodesk Inventor Features

To import features stored in an \*.ipt or \*.iam file that was saved in Inventor while the GibbsCAM CAD Transfer Add-In for Autodesk Inventor was installed, follow these steps. *Before you begin:* On your GibbsCAM workstation, you must have installed either Autodesk Inventor or Autodesk Inventor View, **2014** or later. (Note that Autodesk Inventor View is available as a free download.)

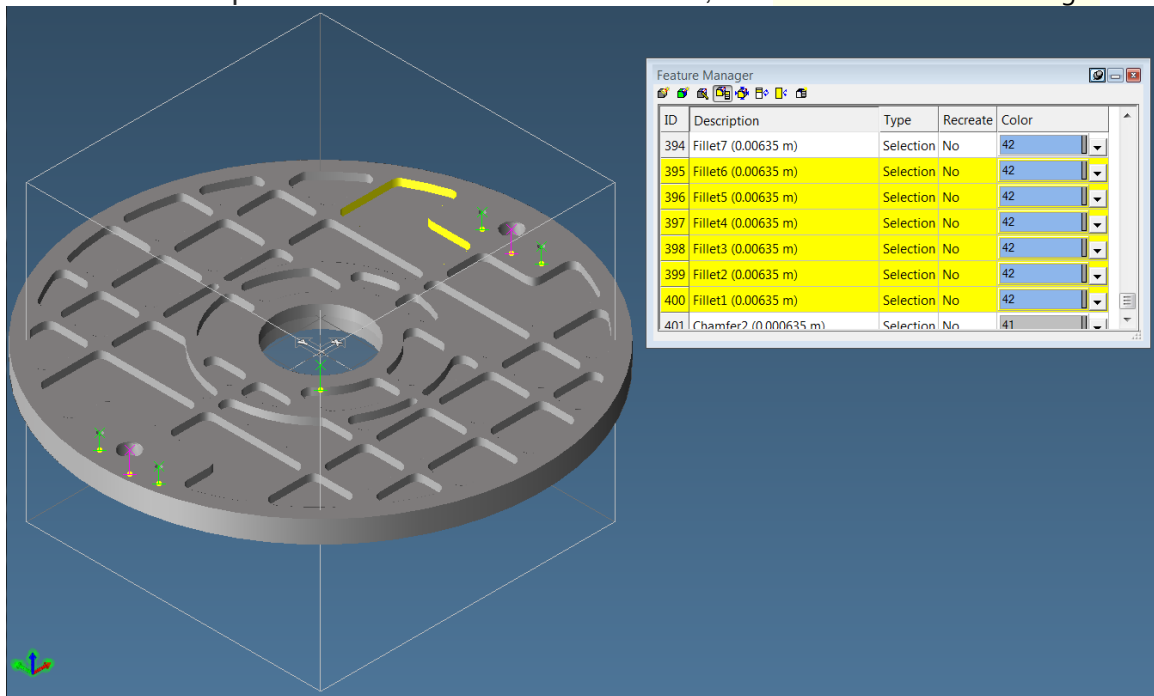
1. **File > Import**; navigate to the Autodesk Inventor file and open it. The following sample file is supplied: **Pockets\_with\_Features.ipt**
2. In the **Solid File Import Options** dialog, specify the settings you prefer.
3. In the information dialog, specify preferences for stitching and units and then click **OK**.

*Result:* The **Define CAD Attribute Map** dialog appears. (Note: This can take several minutes if the file contains many hundreds of features.) If the dialog does not appear, use Plug-Ins menu option Autodesk Inventor Options as noted above to specify importation of features.

In this dialog, specify the types of Inventor features to import, and then click **OK**.



- To view and manipulate feature data within GibbsCAM, use Features > Feature Manager.



## IGES

IGES solid entities are also supported with GibbsCAM. The table shows the list of specific solid entities that can be read by the IGES import filter.

Entity #	Entity Name
186	Manifold Solid B-Rep Object
502	Vertex
504	Edge

Entity #	Entity Name
508	Loop
510	Face
514	Shell

## Surface-Based Files

When surface files are opened/imported, they are displayed as sheets. Sheets are the graphical objects used by the solid modeler to represent surface entities. Each surface entity will be represented as a single faced sheet. When opening/importing surface files an option to solidify the model is available.

## ACIS

### .sat Files

SAT files can be opened directly by GibbsCAM with the SAT (ACIS) option installed; this option is included with 2.5D Solids and above. An initial scan of the entities in the selected file will be made before opening the SAT info dialog. The SAT info dialog provides information on the valid bodies contained in the file.

GibbsCAM will read the SAT file to determine what it is and where it came from. This information is contained within the file's header and will be displayed in the SAT info dialog. Older SAT files do not have this header information. Whatever information that can be determined will be displayed in the dialog. The SAT dialog contains the size specifications for the part model.

SAT files are written in generic units. It is not specified whether these units are mm, inches, miles, etc. The SAT dialog which comes up when a SAT file is imported asks the user to indicate a conversion value for mm per unit. If the units are inches, enter 25.4 mm/unit. Clicking the **in.** button will automatically enter this value in the conversion field.

#### Version

This is the ACIS version that outputs the SAT file.

#### Product

The CAD product used to create the SAT file.

#### ACIS Version

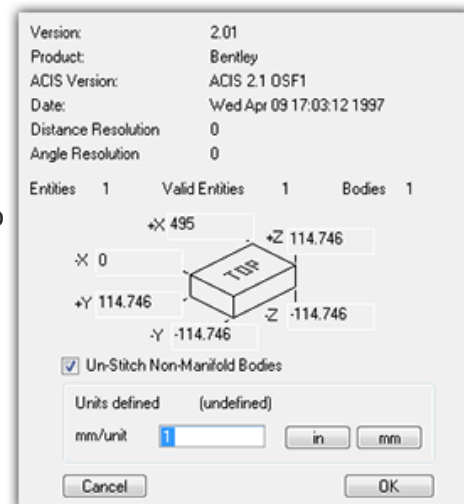
The version of ACIS the file was saved as.

#### Date

The date on which the file was created.

#### Distance Resolution

The distance (not in measurable units) between which any two points are coincident.



**Angle Resolution**

The minimum determinable angle value based on distance resolution and measurement units.

**Entities:**

The total number of objects contained in the file. This can be just bodies or it can be bodies as well as extraneous information.

**Valid Entities:**

The total number of valid entities that GibbsCAM can read. The total may include invalid bodies.

**Bodies**

The total number of bodies that ACIS considers valid.

**Size Specifications for the Part Model**

SAT files are written in generic units. It is not defined whether these units are millimeters, inches, or meters.

**Un-Stitch Non-Manifold Bodies**

This option is available to help importing some files. A body must have two faces connected at every edge. More than two faces at an edge produces a non-manifold solid. By unstitching the body, the file should be able to be imported without error messages. Disable this option if you want to try to import the file directly.

**User Defined Units:**

The user is asked to define a conversion value for mm per unit. If the units are inches enter 25.4 mm/unit. Clicking on the in button will automatically enter this value in the conversion box. Be sure that the unit of measure for a part is the same as that designated in the DCD (Document Control dialog). If you do not know the original units of measure, make an estimate based on the units shown in the dialog. This section will be grayed out if the units are specified in the header.

## ACIS Imports

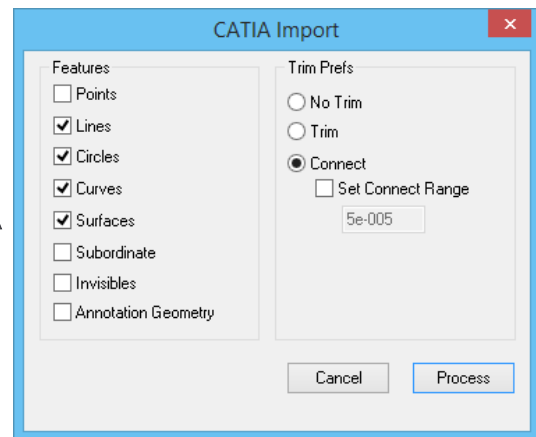
ACIS Mend Body and ACIS Auto HEAL are two preferences that can aid in solidifying imported ACIS files. ACIS Mend Body is active by default and ACIS Auto Heal is not. For most part the default setting or a combination of settings should work well on most parts. If you have an ACIS file that does not import properly, you should try disabling the ACIS Mend Body option. If that also fails, then you should try activating the ACIS Auto Heal option. Note that importing with Auto Heal on can take much more time to import than with it disabled. For more information, see [Common Reference](#).

## CATIA v4

## .dlv and .model Files (Version 4)

If you purchase the CATIA product option, you can import CATIA files into GibbsCAM using **File > Open or File > Import**. When a CATIA file is selected, the **CATIA Import** dialog prompts you to select the features to be imported and to designate Trim Preferences.

Because CATIA files can contain more than one model, the dialog presents a list of models from which you can select. Once a CATIA Model Name is selected, the Reference Coordinate System must be selected. This is the CS in the CATIA file that will correspond to the XY plane in GibbsCAM. Options in the Trim Preferences area control the trimming of wireframe geometry.



## CATIA V5

### .CATPart and .CATProduct Files

CATIA v5 parts are imported without a filter, using **Spatial** (a unit of Dassault Systèmes, and developer of the 3D ACIS kernel).

**Assemblies.** The system can read CATIA v5 Assembly files (CATIA release 7 through release 23). To open a CATIA Assembly, both the assembly and the individual parts it uses must all reside in the same folder.

**Import options.** The import options for CATIA v5 are less numerous than for v4, and you are not prompted for CATIA-specific information.

## IGES Surfaces

IGES surface files can be imported by GibbsCAM. It will attempt to create a valid single faced sheet for each surface contained in the IGES file. However, degenerate surfaces can be created in surface files because they are approximated. When a surface can not be created the imported the ambiguous surfaces will not be created and new surfaces must be created to solidify the model.

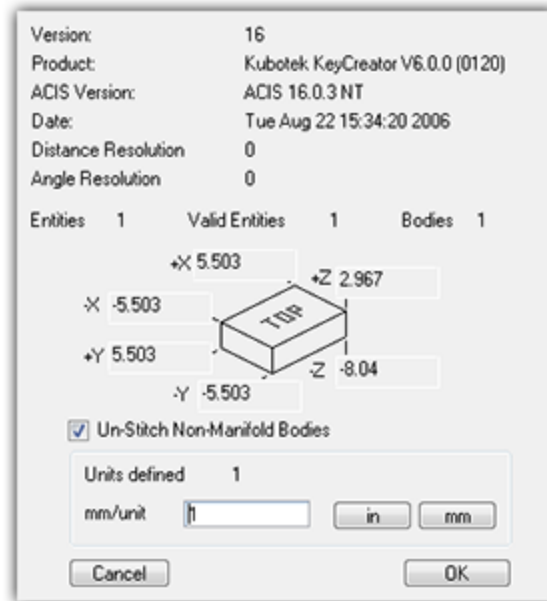
Entity #	Entity Name
114	Parametric Spline Surface Entity
118	Ruled Surface Entity
120	Surface of Revolution Entity
122	Tabulated Cylinder Entity
128	Rational B-Spline Surface Entity
143	Bounded Surface Entity
144	Trimmed Surface Entity

## Errors

GibbsCAM generates an error file when surfaces contained in an IGES file fail to import. An alert will be produced, indicating how many surfaces failed to import and create an error log.

## KeyCreator

### .CKD files



KuboTek KeyCreator files (.CKD files) can be imported when the SAT option is installed. Both individual files and assembly files are supported. When an assembly file is imported, the system will display the part info dialog for each component.

## PTC Creo Parametric (Creo Elements, Pro/E Wildfire)

*Note:* PTC Creo Parametric was formerly known as Creo Elements and as Pro/ENGINEER (informally: “Pro/E”). Its maker, PTC Inc., was once known as Parametric Technology Corporation.

### .prt / .prt\* and .stp / .step Files

You can open and import native PTC Creo Parametric files – in other words, \*.prt and \*.prt\* files. This lets you to open and import STEP files (AP203, AP214, and AP242).

## .asm and .asm\* Files

Although you can import PTC Creo Parametric Assembly files, there are a few things you may need to know in order for it to work as you might expect. The only time you may need this is when you are attempting to import assemblies that contain parts that are non-generic instances in family tables. To retrieve any non-generic instances from a family table, you might requires an instance accelerator file for the instance.

The following information on creating instance accelerator files is adapted from the Help file for Pro/ENGINEER 2001.

## Instance Accelerator Files

You can reduce much of the time required to retrieve an instance of a part or assembly from disk by saving instances in uniquely-named *instance accelerator files*:

- For parts, instance accelerator files are named *instancename.xpr*.
- For assemblies, instance accelerator files are named *instancename.xas*.

The system behavior in saving instances of parts and assemblies is controlled by a variable named *save\_instance\_accelerator*. You can set up or modify the value of the variable either by using a configuration file option or by using the PTC Creo Parametric menu (File > Instance Operations > Accelerator Options) to open the Instance Accelerator dialog box.

The *save\_instance\_accelerator* variable is set to one of the following values:

### None (default)

GibbsCAM saves an instance only by saving the generic model and its Family Table.

### Always

GibbsCAM saves an accelerator file when the instance itself is saved, either explicitly or implicitly:

- The instance is explicitly saved when you open an instance in a new PTC Creo Parametric window using **Open** in the Family Table, and then save the instance as a separate model file using **File > Save** or **File > Save a Copy** in the PTC Creo Parametric menu bar.
- The instance can also be saved implicitly through a higher level object, such as when you save the entire model file using **File > Save** on the PTC Creo Parametric menu bar.

### Explicit

GibbsCAM saves an accelerator file only when the instance itself is explicitly saved. See the preceding paragraph for a description.

In addition to the *None|Always|Explicit* settings (located in the Using Options area) you can also use the Instance Accelerator dialog box to do the following:

- Create an instance accelerator file
- Save instances of parts and assemblies
- Update the generic and all instance accelerator files
- Delete invalid instance accelerator files

## STEP - AP203, AP214, and AP242

STEP files are imported using **Spatial** (a unit of Dassault Systèmes, and developer of the 3D ACIS kernel).

### **.step and .stp Files**

When a STEP assembly file is brought into the system, each component will lie in its own coordinate system so that the imported data looks like the assembly. You can machine the model in the local CS, or you can move the model to a different CS for machining.

## VDA-FS

### **.vda Files**

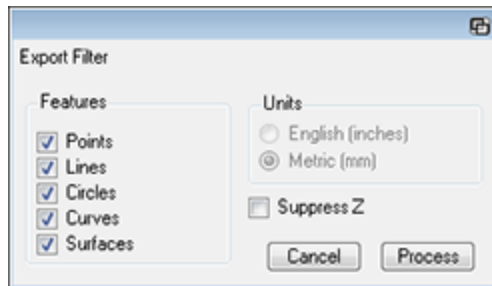
When a VDAFS file is selected and the Solid File Open Options are set, GibbsCAM will immediately open or import the file.

# Export

## Geometry Export

GibbsCAM provides the user with the capability to export geometry contained in a part file into either DXF, IGES or Point list file formats. These files can then be opened by any program that can read those file formats. Files are exported using File > Export. If the file contains multiple workgroups, those workgroups will be split into different layers in the resulting DXF or IGES files. The options include DXF, IGES and Point List file formats. A file name and location to save the file is also entered in this dialog. GibbsCAM will automatically enter an extension onto the file name based on the file format. File extensions are set in the File Extensions preference.

## DXF

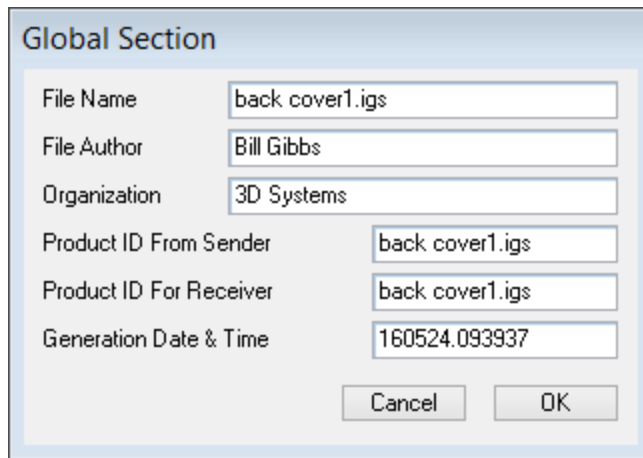


When exporting to the DXF file format, the Export Filter dialog will appear. The Features group designates what features to export to the DXF file. If the Suppress Z button is turned on, the exported DXF file will not contain any Z values, producing a two-dimensional drawing. Many CAD packages that use the DXF file format only support 2D drawing of geometry. The Units box is grayed out when exporting to the DXF file format. DXF files do not

make any designation to units of measurement. GibbsCAM will output the geometric values in whichever working units are being used in the file. GibbsCAM imports the DXF entities shown in the following table.

Entity	Feature Equivalent
POINT	Point
LINE	Line
CIRCLE	Circle
POLYLINE	Connected Lines

# IGES



**Global Section**

File Name: back cover1.igs

File Author: Bill Gibbs

Organization: 3D Systems

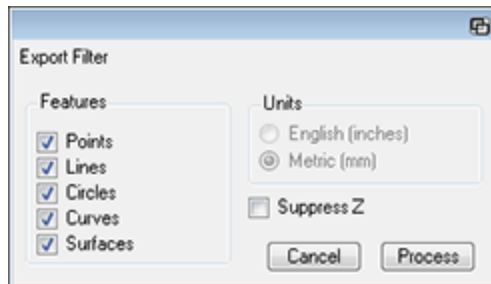
Product ID From Sender: back cover1.igs

Product ID For Receiver: back cover1.igs

Generation Date & Time: 160524.093937

Cancel OK

GibbsCAM can export geometry and solids as IGES surface entities. When exporting to the IGES format file format (once the file has been named and saved), the dialog shown to the right will come up. The information can be altered as needed by the user. This dialog contains header information that will be written to the IGES file. The file name will be entered as it was in the Export dialog and the information will default to previous entries made in those fields. The Generation Date & Time is set according to the date and time on the computer.



**Export Filter**

**Features**

- ☒ Points
- ☒ Lines
- ☒ Circles
- ☒ Curves
- ☒ Surfaces

**Units**

☐ English (inches)

☒ Metric (mm)

☐ Suppress Z

Cancel Process

Once the header information has been entered, the Export Filter dialog shown to the right will come up. This dialog controls how the file will be created. The Features group allows the user to select which features to export. Points, lines, circles and curves can be exported. The Units group allows parts to be exported in inches or metric units. The Suppress Z checkbox is available only when exporting DXF files.

IGES is supported by many CAD systems and is published by the National Bureau of Standards, US Department of Commerce. The IGES entities shown in the following table can be exported by GibbsCAM.

Entity #	Entity Name	Feature Equivalent
100	Circular Arc Entity	Circle
110	Line Entity	Line
116	Point Entity	Point
124	Transformation Matrix Entity	Coordinate System Information
126	Rational B-Spline Curve Entity	Connected lines

## Point List

When exporting a point list, only the name and location to save the file need to be specified. To create the point list text file, click on the Save button. The resulting text file will contain the X, Y, and Z coordinates of any points contained in the current file. Each point will be on a separate line and each coordinate value will be separated by one space.

# Solid Export

GibbsCAM can write files into different file formats including Parasolid, ACIS, and STL. In many cases files can be exported to various versions of many types of formats. Each export will display its version in the **Save as type** list as shown. To save the file to the latest version use the entry with no version number listed.

## ACIS

### .sat Files

GibbsCAM supports multiple versions of ACIS (.SAT) files starting with v6. You can save a model to older ACIS versions to be compatible with older CAD systems. To save the file to the most recent version of SAT, use the **ACIS-SAT (\*.sat)** entry with no version number listed.

## Parasolid

### .x\_t Files

Both sheets and solids can be exported into Parasolid files. Parasolid can be exported to many versions. GibbsCAM is based on the current Parasolid technology. When the standard **Parasolid (\*.x\_t)** item is selected for the file type, the resulting file will be based on the current version of Parasolid. Older versions display their version numbers. These selections can be used to export a Parasolid file that will be compatible with a system using older Parasolid technology. If you export a facet body to Parasolid, *two* files will be created: *filename.x\_t* and *filename.x\_t\_fbi*.

## Stereolithography

### .stl Files

An STL file is for use with stereolithography systems. An STL file is a faceted model, not a surface model. The file contains no surface information, only the faceting information necessary to represent or render the model.

**ASCII / Binary.** ASCII files are nearly four times larger than binary files, but are more portable and more human-readable.

**Regular / Plus.** When you export to **STL [ASCII]** or **STL Binary**, the faceting tolerance is determined by the chord height set for the body in its Properties dialog. When you export to **STL Plus [ASCII]** or **STL Plus Binary**, you specify facet accuracy as you export.

## Neutral-Format (ATB) for PTC Creo Parametric (Creo Elements, Pro/E Wildfire)

## .neu Files

*Note:* PTC Creo Parametric was formerly known as Creo Elements and as Pro/ENGINEER (informally: “Pro/E”).

You can export solids to *Neutral* format, also called ATB (for Associative Topology Bus).

## STEP - AP203, AP214, and AP242

### .step and .stp Files

You can export solids to the STEP (AP203, AP214, and AP242) formats.

# Transfer Add-Ins

## CAD Interoperability

GibbsCAM provides for several CAD Transfer Add-Ins. These Add-Ins provide interoperability between: GibbsCAM and Autodesk Inventor, GibbsCAM and KeyCreator, GibbsCAM and Rhinoceros, GibbsCAM and Solid Edge, GibbsCAM and CimatronE, and GibbsCAM and Solidworks. The GibbsCAM Transfer Add-In creates a menu item in your CAD application that allows for a part to be directly transferred to GibbsCAM. The Add-In maintains associativity between the native model and toolpath generated in GibbsCAM. Associativity will remain as long as the model has not been changed within GibbsCAM.

## Common Functionality

When a part is transferred, a new file will be created with the same name as the original part. GibbsCAM will automatically launch if it is not currently running. If GibbsCAM is running and has an open part file you will be prompted with the options **New Part**, **Insert** and **Replace**.

### **New Part**

**New Part** will close the current GibbsCAM file and create a new part file with the new part.

### **Insert**

**Insert** will add the new part to the current GibbsCAM file.

### **Replace**

**Replace** will replace all bodies that are designated as a “Part” in the Properties dialog with the new part. Bodies that are designated as “Stock” or “Fixture” are not replaced.

Note: In some rare instances, GibbsCAM may display an error upon launching and the part will not be transferred. If this happens, click **OK** on the alert, and retry the transfer. Also, some types of geometry edits may invalidate the associativity, (e.g. topological changes).

## Autodesk Inventor Transfer Add-In

The GibbsCAM Autodesk Inventor Transfer Add-In lets you transfer Autodesk Inventor file types for parts (\*.ipt) and assemblies (\*.iam). Only one file type can be transferred at a time. When transferring, all sub-assemblies are dissolved into the main assembly. After bodies are transferred to GibbsCAM, they are separate solids that can be handled individually. The CAD Transfer Add-In for Autodesk Inventor must be installed on the system where Autodesk Inventor is installed.

### **Use**

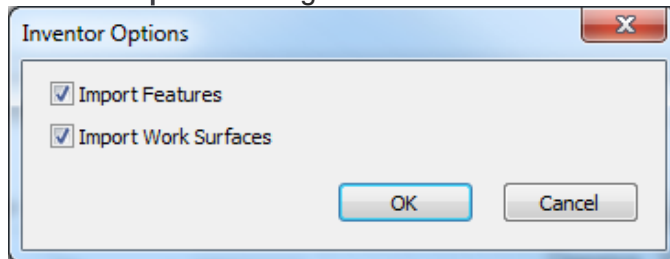
The Add-In is accessed from Inventor's main menu: Add-Ins > Transfer To GibbsCAM.

You can store Autodesk Inventor features in a way that makes them understandable to GibbsCAM.

## What Are Autodesk Inventor Features?

Autodesk Inventor defines several kinds of features: holes, extrusions, fillets, chamfers, and so forth. Features in Autodesk Inventor part files (\*.ipt) and assembly files (\*.iam) can be understood by GibbsCAM 2026, but only if they were saved while the GibbsCAM CAD Transfer Add-In for Autodesk Inventor was installed on the Autodesk workstation.

Such feature data can sometimes be quite large, and it is not always needed. In GibbsCAM, you can optionally set a switch that suppresses the importation of such feature data, using the **Inventor Options** dialog box:



To open this dialog box: Plug-Ins > (Misc) > Autodesk Inventor Options

## Importing Autodesk Inventor Features

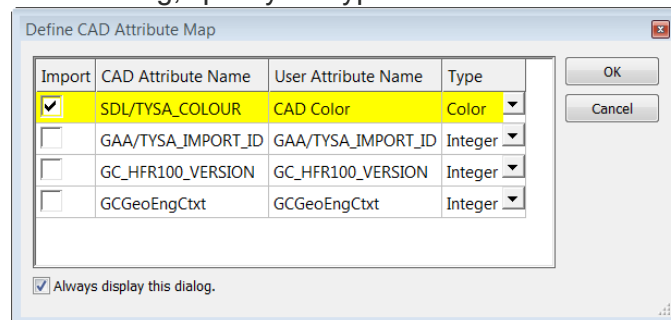
To import features stored in an \*.ipt or \*.iam file that was saved in Inventor while the GibbsCAM CAD Transfer Add-In for Autodesk Inventor was installed, follow these steps.

*Before you begin:* On your GibbsCAM workstation, you must have installed either Autodesk Inventor or Autodesk Inventor View, 2014 or later. (Note that Autodesk Inventor View is available as a free download.)

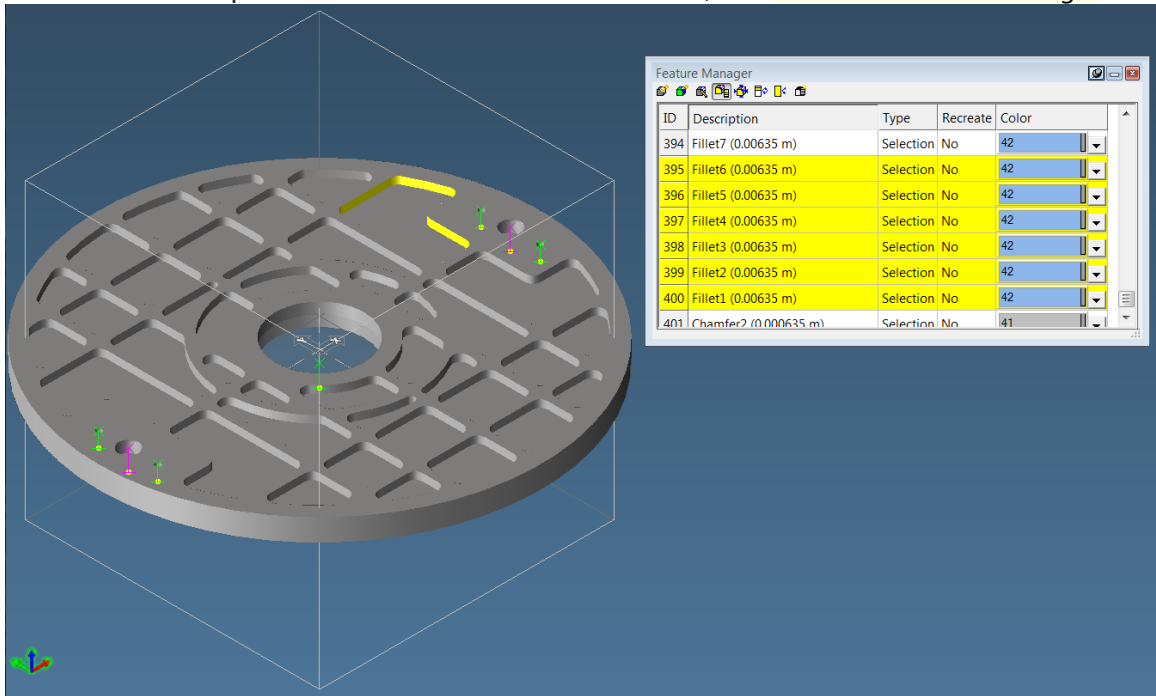
1. File > Import; navigate to the Autodesk Inventor file and open it. The following sample file is supplied: **Pockets\_with\_Features.ipt**
2. In the **Solid File Import Options** dialog, specify the settings you prefer.
3. In the information dialog, specify preferences for stitching and units and then click **OK**.

*Result:* The **Define CAD Attribute Map** dialog appears. (Note: This can take several minutes if the file contains many hundreds of features.) If the dialog does not appear, use Plug-Ins menu option **Autodesk Inventor Options** as noted above to specify importation of features.

In this dialog, specify the types of Inventor features to import, and then click **OK**.



5. To view and manipulate feature data within GibbsCAM, use Features > Feature Manager.



## KeyCreator Transfer Add-In

The GibbsCAM KeyCreator (v6.3 and above) Transfer Add-In will transfer design (CKD) KeyCreator file types. Only one file type may be transferred at a time. This add-in process needs to be repeated with each launch of KeyCreator.

### Use

To transfer a KeyCreator part choose Add-Ins > CDE open. Then browse to the GibbsCAM installation folder and open the KeyCreator folder, select KeyCreator Addin.CDE and Open. Now click the Transfer button. The model will rotate to a top view and transfer. The transfer can also be accessed from Add-Ins > GibbsCAM > Transfer > GibbsCAM Transfer located your KeyCreator menu bar.

## Rhinoceros Transfer Add-In

The GibbsCAM Rhinoceros Transfer Add-Ins will transfer design (3DM) Rhinoceros 3.0, 4.0 and 5.0 file types. Only one file type may be transferred at a time. This add-in process needs to be repeated with each launch of Rhino.

### Use

To transfer a Rhinoceros part choose Tools > Options. Then select Plug-ins from the Options list and click the Install button. Browse to the C:\Program Files\Common Files\GibbsCAM Shared\Interchange\ folder, open the Rhino5 folder and select GibbsCAM.RHP. Click the selection Open > GibbsCAM > Transfer Part To GibbsCAM that is now available in your Rhino menu bar.

## Solid Edge Transfer Add-In

The GibbsCAM Solid Edge Transfer Add-In allows you to transfer part (PAR), draft (DFT), assembly (ASM), and sheet metal (PSM) Solid Edge file types. Only one file type may be transferred at a time.

### Use

Within the Solid Edge application, use menu option Add-Ins > Transfer Part To GibbsCAM.

## SolidWorks Transfer Add-In

The GibbsCAM SolidWorks Transfer Add-In will transfer SolidWorks file types for parts (\*.SLDPRT), assemblies (\*.SLDASM), and drawings (\*.SLDDRW). Only one file type can be transferred at a time. When transferring, all sub-assemblies are dissolved into the main assembly. After bodies are transferred to GibbsCAM, they are separate solids that can be handled individually.

The CAD Transfer Add-In for SolidWorks must be installed on the system where SolidWorks 2011 or later is installed. SolidWorks 2011 (or later) is required for the Add-In to work.

### Use

The SolidWorks Transfer Add-In is accessed from GibbsCAM Transfer Add-In > Transfer Part To GibbsCAM.

### Installing the Add-In on Workstations Without GibbsCAM

If you need to install the SolidWorks Add-In on a workstation that does not run GibbsCAM, run the standalone installer (\*.msi file) provided on the GibbsCAM installation DVD (under Content\Tools\).

If you have a Gibbs Online Services account, you can download a 32-bit or 64-bit version of this standalone installer from <https://online.gibbscam.com> : Software Downloads > Tools. The MSI file can be freely distributed; however, be careful to review the license agreement.

## CimatronE Transfer Add-In

The GibbsCAM Cimatron Add-In allows direct transfer of CimatronE files into GibbsCAM. You can either transfer the entire screen display or selected entities.

### Installation

When installing a newer version of GibbsCAM it is important to select the newest version of CimatronE.

If you are running an older version of GibbsCAM with CimatronE already installed, you need to first uninstall CimatronE (Change installation) before installing the latest version in order for GibbsCAM to take advantage of the new CimatronE Add-ins.

### Use

The Add-In is accessed from GibbsCAM > Transfer/GibbsCAM Transfer Selected.

# Conventions

GibbsCAM documentation uses two special fonts to represent screen text and **keystrokes or mouse actions**. Other conventions in text and graphics are used to allow quick skimming, to suppress irrelevancy, or to indicate links.

# Text

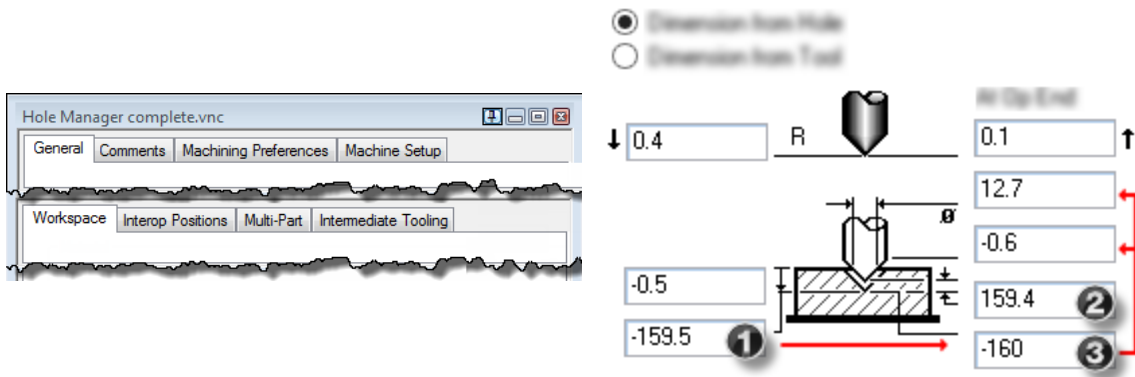
Screen text. Text with this appearance indicates text that appears in GibbsCAM or on your monitor. Typically this is a button or text for a dialog.

Keystroke/Mouse. Text with **this appearance** indicates a keystroke or mouse action, such as **Ctrl+C** or right-click.

**Code.** Text with **this appearance** indicates computer code, such as lines in a macro or a block of G-code.

# Graphics

Some graphics are altered so as to de-emphasize irrelevant information. A “torn” edge signifies an intentional omission. Portions of a graphic might be blurred or dimmed to highlight the item being discussed. For example:



Annotations on a graphic are usually numbered callouts (as seen above), and sometimes include green circles, arrows, or tie-lines to focus attention on a particular portion of the graphic.

# Links to Online Resources

Please contact your reseller for support.

Link	URL	Action / Description
<a href="#">Go</a>	<a href="http://www.GibbsCAM.com">http://www.GibbsCAM.com</a>	Opens the main website for GibbsCAM.
<a href="#">Go</a>	<a href="https://online.gibbscam.com">https://online.gibbscam.com</a>	Opens Gibbs Online page to download GibbsCAM and all supported material.

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