Pro/ENGINEER®
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Pro/INTERFACE™
Help Topic Collection

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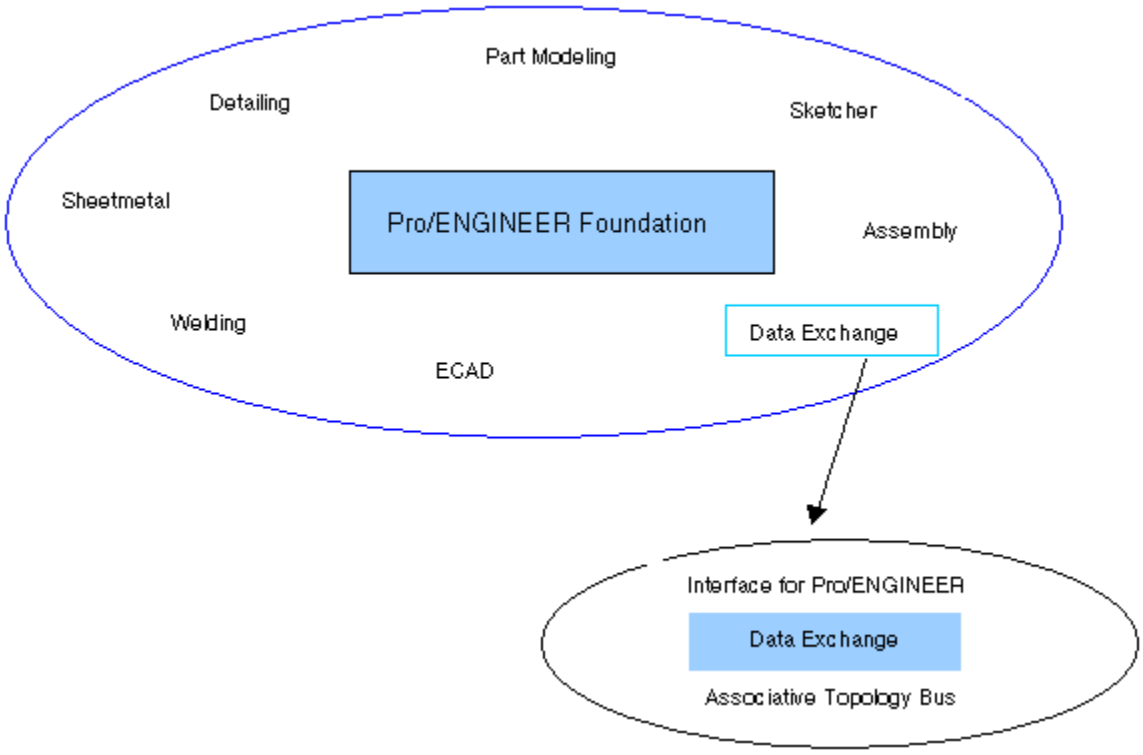
Pro/ENGINEER Interface

About Data Exchange in Pro/ENGINEER

Using direct data exchange and industry-standard translators, you can share and reuse engineering data by importing geometry to Pro/ENGINEER and exporting geometry from Pro/ENGINEER. Pro/ENGINEER has direct geometry translators for ACIS, Adobe Illustrator, CADDS 5, CATIA, CDRS, ICEM, I-DEAS, Parasolid, PDF, Pro/PHOTORENDER, CADAM, MEDUSA (3D ASCII format), Unigraphics, and AutoCAD DXF/DWG.

Industry-standard translators in Pro/ENGINEER support the following data formats: IGES, STEP (AP202, AP203, AP214 - including Associative Drafting), SET, VDA, ECAD (IDS 2.0, 3.0), CGM, COSMOS/M, PATRAN and SUPERTAB geometry files, SLA, CGM (MILSPEC MIL-D-28003A), JPECT, TOFF, ProductView, RENDER, STL, VRML, INVENTOR, ACIS, STHENO/PRO, and XPATCH.

Where Does Interface Fit in Pro/ENGINEER?

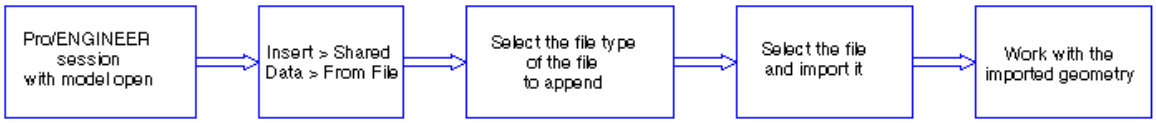


Workflows

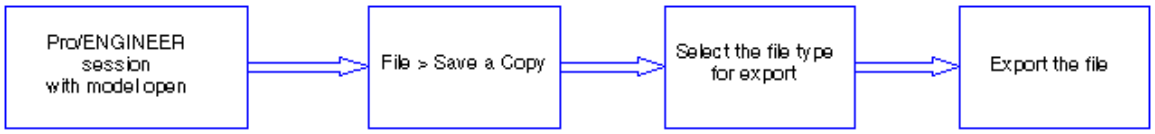
Basic workflows for data exchange are as follows.

Importing a Model to Pro/ENGINEER

Appending a Model to a Pro/ENGINEER Model



Exporting a Pro/ENGINEER Model



Using Pro/ENGINEER Interface

About Interface Capabilities

The Interface for Pro/ENGINEER is part of Pro/ENGINEER Foundation. Capabilities of the Interface for Pro/ENGINEER include capabilities of the following data exchange formats.

ACIS

You can:

- Export and import parts and assemblies in the ACIS (`.sat`) file format.
- Open the exported parts and assemblies in MECHANICA.

Adobe Illustrator Curves

You can:

- Import Adobe Illustrator curves directly into Pro/ENGINEER as parts, assemblies, or sketches.
- Modify the curves that are imported as features to create other geometry.

BATCH

You can enter Batch mode to create multiple plot, IGES, DXF, VDA, SET, STEP, STL, or VRML files.

CADAM/CDT/CPTR

You can:

- Import and modify a drawing directly from the Professional CADAM database through Pro/CDT utilities (CADAM Data Transfer).
- Import CADAM mainframe files through the CPTR (CADAM to Pro/ENGINEER Transfer) neutral file (`.bin`) format.

CADDS 5

You can:

- Import CADDS parts and assemblies into Pro/ENGINEER.
- Export Pro/ENGINEER parts and assemblies to native CADDS format.

For Associative Topology Bus (ATB)-enabled CADDS 5, see the Help on Associative Topology Bus.

CATIA

Pro/ENGINEER supports CATIA versions 4 and 5. You need separate licenses for data exchange between Pro/ENGINEER and each of these versions of CATIA. You can export and import both part and assembly models between Pro/ENGINEER and CATIA version 4. You can only import CATIA version 5 part models to Pro/ENGINEER.

You can export and import to Pro/ENGINEER the following file types:

- CATIA V4 `.model`
- CATIA V4 ProGEO `.model`
- CATIA V4 Model export (`*.exp`)
- CATIA V4 Session (assembly) `*.session`
- CATIA V5 CATPart (part), revision 6 and later

CATIA is an ATB-enabled application. To know more about the ATB capabilities, see the Help on Associative Topology Bus.

CDRS

You can exchange data between CDRS and Pro/ENGINEER using **File** > **Open** or **Insert** > **Shared Data** > **From File**. Industrial design surfaces are created in CDRS and imported to Pro/ENGINEER for downstream and structural design and creation of engineering deliverables. For information on ATB-enabled CDRS, see the Help on Associative Topology Bus.

Computer Graphics Metafile (CGM)

CGM provides a vector-based 2D image file format for the storage and retrieval of graphics information.

You can:

- Export graphical information to the CGM format in Part, Assembly, and Drawing modes.
- Import a CGM file into a drawing, format, layout, or diagram.

COSMOS

COSMOS/M files are ASCII text files. You can:

- Use COSMOS geometry to create a neutral file that can be read by COSMOS/M (COSMOS/M is the name of a FEM solver from SRAC).
- Rename COSMOS files and modify them using standard operating system commands.

Data Exchange Format (DXF) and DWG

You can import a DXF file and modify the resulting drawing or create design models and construct features. A DXF file can contain 2D or 3D geometry. You can:

- Import and export 2D DWG files to and from products such as AutoCAD.
- Use the DXF files containing 3D geometry to import parts, assemblies, components, and features. Tessellated data and embedded exact ACIS data contained in the DXF files are also imported.

Electrical Computer-Aided Design (ECAD)

You can use the **ECAD** option to exchange information between Pro/ENGINEER and ECAD systems.

ICEM

You can create Pro/ENGINEER models by importing ICEM Surf models. ICEM surface, when imported into Pro/ENGINEER or appended to models in Pro/ENGINEER, create import IN features.

I-DEAS

You can import to Pro/ENGINEER I-DEAS file types `.mf1` and `.pkg` and selectively filter the part and assembly design models contained in these files during import.

This interface is not included in Pro/ENGINEER Foundation and requires a licensed installation. Pro/ENGINEER supports I-DEAS version 10.0.

Initial Graphics Exchange Specification (IGES)

You can:

- Export drawing, drawing format, layout, part, and assembly data in IGES format.
- Export part data using B-Spline representation for all surfaces and automatically initiate post-processing of the IGES file using a program that you supply.
- Import IGES files containing drawing data into drawings, formats, and layouts and modify the resulting product.
- Group model edges in a drawing for export through the IGES format to allow other systems that support IGES groups to see the model edges as a single collection of entities.
- Import IGES files containing drawing, sketch, and part data into all modes of Pro/ENGINEER.

Image Files

You can:

- Print shaded images to a PostScript printer.
- Export TIFF, JPEG, and EPS files.
- Import and export CGM picture files.
- Import TIFF images into Drawing mode.

MEDUSA

You can:

- Export Pro/DETAIL drawings to MEDUSA as MEDUSA sheets.
- Import MEDUSA sheets into Pro/ENGINEER.
- Export solid model information about parts and assemblies to MEDUSA as tessellated data. MEDUSA exports to an ASCII-based `*.asc` file.
- Import tessellated MEDUSA `*.asc` files.

Neutral Files

You can use:

- The **Neutral** option to create or import a formatted text file (NEUTRAL file) containing information such as model topology, relations, and attributes about parts and assemblies created with Pro/ENGINEER.
- The NEUTRAL file to create interfaces with other programs.
- The Neutral file format to collaboratively share data created in Pro/ENGINEER with the earlier versions of Pro/ENGINEER.

The Neutral file format is ATB-enabled. For information, see the Help on Associative Topology Bus.

Parasolid

You can export Pro/ENGINEER parts and assemblies to Parasolid and import 3D geometry into Pro/ENGINEER from a Parasolid-based CAD system in the Parasolid format.

PATRAN

You can export a file from part data that can be read by PATRAN software systems. This file is formatted according to the specifications of a PATRAN database file. The PATRAN file contains mathematical definitions of items such as surface data that bounds a solid. Pro/ENGINEER does not accept files from PATRAN systems as input.

Portable Document Format (PDF)

You can export 2D Pro/ENGINEER drawings, formats, and layouts directly to PDF using Adobe Distiller.

Note: Pro/ENGINEER supports PDF on the Windows platform only.

PHOTORENDER

You can create photorealistic images of Pro/ENGINEER, Pro/NOTEBOOK, and Pro/CDRS models through the Neutral file format.

Pro/DESKTOP

You can open in Pro/ENGINEER Pro/DESKTOP native `.des` part files of Pro/DESKTOP V7 and higher and native `.des` assembly files of Pro/DESKTOP V8 and higher. You can also open the Pro/DESKTOP to Pro/ENGINEER export format, `.pdt` files, from any version of Pro/DESKTOP. The `.pdt` format only supports parts. Pro/DESKTOP `.pdt` files, version 2001 or higher, support Conceptual Engineering Data (CED) transfer.

Because native Pro/DESKTOP V7 and higher `.des` files are Granite-based, you can open them with the Granite 'direct open' method which is the default or the standard import method.

Pro/DESKTOP is an ATB-enabled format. For more information, see the Help on Associative Topology Bus.

ProductView

You can:

- Export a part, assembly, or drawing to the ProductView format.
- Import from ProductView facet parts and assembly structures.

Pro/ENGINEER supports ProductView version 6.0.

Scan Data

You can digitize from a tablet to create point, line, and arc data from existing drawings and layouts for import into Pro/ENGINEER.

Standard for Exchange and Transfer (SET)

You can:

- Import and modify a drawing that contains SET files.
- Import SET part and drawing files, including formats and layouts.
- Export SET part data (curve and surface data and coordinate systems) and SET drawing data (2D geometry, text, and dimensions).

Shrinkwrap

You can give each Shrinkwrap model a name and store it as a separate Pro/ENGINEER part.

STEP

You can:

- Exchange complete product definition between heterogeneous computer-aided design, engineering, and manufacturing systems through the Standard for the Exchange of Product Model Data (STEP) format.
- Import and export STEP Associative Drafting Data (AP214).

STHENO/PRO

STHENO/PRO provides early 2D concept sketching and extensive capabilities for production detailing of drawings created in Pro/ENGINEER. In the integrated mode of STHENO/PRO, you can:

- Alternate or work with multiple Pro/ENGINEER drawing sheets.
- Transfer Pro/ENGINEER drawing information to STHENO/PRO.
- Add information and still return to the original sheet of a Pro/ENGINEER drawing for further modification.

You can also exchange data between Pro/ENGINEER and STHENO/PRO in the `.tsh` file format through the export, import, and append operations.

Supertab

You can export an I-DEAS Supertab file from part data that can be read by Supertab software systems. This file is formatted according to the specifications of a Supertab universal file. The Supertab file contains mathematical definitions of items such as surface data that bound a solid. Pro/ENGINEER does not accept universal files from Supertab systems.

Tessellated Data

You can create tessellated files by exporting to STL, Render, OptegraVis, Xpatch, CatiaFacets (Catia Mock-Up), MEDUSA, Pro/CONCEPT, or Inventor. The dialog box that opens before you export your files allows you to specify tessellation parameters. You can:

- Import and export STL files that enable Pro/ENGINEER part and assembly geometry to be read by stereolithography or rendering programs. STL files represent the surfaces of a solid model as a group of small planar polygons.
- Create a RENDER file that enables Pro/ENGINEER part and assembly geometry to be read by rendering programs. RENDER files contain curved surface element definition and color to produce high-quality shaded images. These files are incompatible with stereolithography programs.

- Export part and assembly information that can be read by IRIS Inventor, a 3D graphics toolkit by Silicon Graphics.
- Export the geometry of part and assembly models for use with Pro/CONCEPT through the Wavefront `.obj` files.
- Export a tessellated file to use in Optegra Visualizer software.
- Import and export tessellated MEDUSA `.asc` files.
- Import and export files in VRML format.
- Import and export files in Catia Mock-Up format.
- Export a tessellated model for use in the Xpatch Radar Cross Section analysis software.

TIFF

TIFF supports the interface of digital image data between systems with different architectures. You can use TIFF to store graphical and textual information in a bitmap format and later exchange it between Pro/ENGINEER and different application programs on X Windows on UNIX or GDI on Windows NT.

Pro/ENGINEER supports the export and import of TIFF in all graphics modes.

Unigraphics

You can exchange data between Unigraphics and Pro/ENGINEER through the file import, export, append, and assemble operations. The data can consist of solid geometry, quilts, layers, colors, assembly structures, and units.

Pro/ENGINEER supports Unigraphics Release 18.0 and 19.0 (NX). The Unigraphics interface is not included in Pro/ENGINEER Foundation and requires a licensed installation. You must use the `intf3d_ug_install_dir` configuration option to identify the location of the Unigraphics installation.

Unigraphics is an ATB-enabled file format.

VDA

You can transfer part geometry between computer systems by importing and exporting through the VDA Surface Data Interface available with Pro/ENGINEER Interface.

Note: Pro/ENGINEER only supports the default fonts in 2D import and export for most formats. In DWG and DXF formats, Pro/ENGINEER supports the default fonts for export and the default and IGES-1003 fonts for import to DWG files.

Pro/ENGINEER supports VDA Version 2.0.

Configuring Pro/ENGINEER Interface

About Configuring Pro/ENGINEER Interface

You can customize the way Pro/ENGINEER Interface operates for data exchange by specifying `config.pro` configuration file options and their values in the **Options** dialog box (**Tools** > **Options**).

Pro/ENGINEER Interface Help provides a list of configuration options arranged in alphabetical order. Each option contains the following information:

- Configuration option name.
- Default and available variables or values. All default values are in italic.
- Brief description and notes describing the configuration option.

To Set Configuration Options for Pro/ENGINEER Interface

1. Click **Tools** > **Options**. The **Options** dialog box opens.
2. Click the **Show only options loaded from file** check box to see currently loaded configuration options or clear this check box to see all configuration options.
3. Select the configuration option from the list or type the configuration option name in the **Option** box.
4. In the **Value** box type or select a value.

   Note: The default value is followed by an asterisk (*).
5. Click **Add/Change**. The configuration option and its value appear in the list. A green status icon confirms the change.
6. When you finish configuring, click **Apply** or **OK**.

Note: It is recommended that you set the configuration options for a particular file format before starting a data exchange operation.

About Configuring the Export of Pro/ENGINEER Models to IGES, Parasolid, and STEP

You can use separate configuration files to customize the export of Pro/ENGINEER models to the IGES, Parasolid, and STEP formats. Set up configuration options that are specific to the default export setup for IGES, Parasolid, or STEP in the `iges_config.pro`, `parasolid_config.pro`, or the `step_config.pro` configuration files, respectively.

These configuration files must exist in the Pro/ENGINEER startup directory so that Pro/ENGINEER uses the settings in these files during the export of Pro/ENGINEER models to the IGES, Parasolid, and STEP formats.

Use **Options** on the **Export IGES**, **Export Parasolid**, or **Export STEP** dialog boxes to change the configuration option settings in the `iges_config.pro`,

parasolid_config.pro, or the step_config.pro configuration files, respectively, at runtime. To see the valid options in these configuration files, press F4 in a cell of the first column of the Pro/TABLE. Modify the configuration options in the dialog box that opens.

You can also click **Find** in the **Options** dialog box and use the **Find Option** dialog box to see the valid options. Access the **Options** dialog box by clicking **Tools** > **Options**.

acis_out_version

4, 5

Controls the version of the ACIS file when you export a Pro/ENGINEER part or assembly to the ACIS format.

adobe_distiller

<adobe distiller command path>

Sets the path to the Adobe Distiller executable file. By default, the executable file is acrodist.exe. For example, the value can be c:\program files\adobe\acrobat 6.0\distillr\acrodist.exe.

allow_4dnav_export

yes, *no*

yes—Exports to VRML consistent with CATIA's 4D Navigator.

atb_ident_cadds_files

yes, *no*

Controls the display of CADDS models as file or directory icons in the **File Open** dialog box.

Note: CADDS 5 models are directories and not individual files.

- no—Displays CADDS models as folder icons in the **File Open** dialog box for **All Files (*)** selected in the **Type** box. For **CADDS5** selected as the file type in the **Type** box, CADDS 5 models are displayed as file icons. You can open CADDS 5 files only when you explicitly set the file type as **CADDS5**.
- yes—Displays CADDS 5 models as file icons for both **CADDS5** and **All Files (*)** selected as the file type in the **Type** box. You can open CADDS 5 files for any of these filters. CADDS 5 models are not displayed when you set other format-specific filters.

auto_associate_dimensions

yes, *no*

Associates imported DXF, DWG, and IGES dimensions with the corresponding imported geometry when set with the `associative_dimensioning` drawing setup option set to `yes`.

This configuration option is available as the **Create Associative Dimensions** option in the **Import DXF**, **Import DWG**, and **Import IGES** dialog boxes.

cadam_line_weights

`light`, `medium`, `heavy`

Defines the line width of entities in Pro/ENGINEER to plot drawings with line weights that are consistent. The default values for these weights are:

- `Light`—.2
- `Medium`—.3
- `Heavy`—.5

Set the configuration option as shown in the example to plot a drawing with imported CPTR data and use default line weights:

```
cadam_line_weights .2 .3 .5
```

cadds_import_layer

`yes`, `no`

Maps CADDS 5 layers to Pro/ENGINEER layers during import from CADDS 5 to Pro/ENGINEER.

catia_out_to_existing_model

`append`, `overwrite`

Controls conflict resolution if the selected CATIA model already exists.

- `append`—Appends the exported data to the existing CATIA file.
- `overwrite`—The exported file overwrites the existing CATIA file.

cdt_transfer_details

`no`, *`yes`*

Determines the dittos in a CADAM drawing when importing the CADAM drawing.

- `no`—Places details (dittos) associated with the CADAM drawing on the current Pro/ENGINEER drawing sheet. No extra sheets are added.
- `yes`—Converts details associated with the CADAM drawing to additional sheets on the Pro/ENGINEER drawing. The drawing has as many sheets added as dittos transferred.

cgm_inc_pad_byte_in_length

yes, *no*

Enables a metafile to be processed by the Micrographic CGM converter.

cgm_use_enum_in_real_spec

yes, *no*

Enables a metafile to be viewed in Advanced Technology Center's For Review.

cgm_use_reversed_ieee_floats

yes, no

Enables a metafile to be viewed in Advanced Technology Center's For Review.

copy_dxf_dim_pict

yes, *no*, as_symbol

Determines the import of dimension definitions. This configuration option is available as **Import Dimensions** in the **Import DXF** and **Import DWG** dialog boxes.

- yes—Ignores the dimension definition and only imports the blocks as separate entities. The dimensions look like dimensions in AutoCAD.
- no—Creates dimensions in Pro/ENGINEER according to DIMENSION entity and the entities from the block.
- as_symbol—Ignores the dimension definition and imports blocks as Pro/ENGINEER symbols. The dimensions look like dimensions in AutoCAD.

direct_vrml

yes, no

Controls the appearance of **VRML Converter** on the **File** menu.

- yes—**VRML Converter** appears on the **File** menu, allowing you to use the Direct VRML export functionality.
- no—**VRML Converter** does not appear on the **File** menu.

dwg_export_format

12, 13, 14, *2000*

Enables you to select the DWG file version to which you want to export a Pro/ENGINEER drawing.

dxf_block_to_pro_symbol

yes, *no*

Controls the import of blocks and block instances. Block instances are imported as separate symbols. This configuration option is available as the **Import Blocks As Symbols** option in the **Import DXF** dialog box.

yes—Imports blocks as Pro/ENGINEER symbols.

dxf_export_format

12, 13, 14, *2000*

Enables you to select the DXF file version to which you want to export a Pro/ENGINEER drawing.

dxf_export_mapping_file

Specifies the DXF export mapping file. Specify the absolute or relative path to the file.

dxf_in_faceted_brep

yes, no

Determines the type of geometry that is imported from a 3D DXF file.

- yes—Imports the geometry of the 3D DXF file as faceted or ACIS exact data.
- no—Imports the faceted wireframe 3D geometry in the DXF file.

dxf_out_comments

yes, no

Controls the creation of comment lines in the DXF file.

yes—Creates comment lines in the DXF file.

dxf_out_drawing_scale

yes, *no*

Specifies whether to export the drawing scale to the DXF or DWG file.

- yes—Includes a scale factor, DIMLFAC, in the exported DXF or DWG file to be read by AutoCAD.
- no—Exports without using a scale for the drawing.

dxf_out_scale_views

yes, *no*

Specifies whether to scale drawing views when exporting to the DXF or DWG file format.

- yes—Rescales the entire drawing such that the main view scale is 1:1 before export.

- no—Exports without rescaling the drawing.

dxf_out_sep_dim_w_breaks

yes, *no*

Determines how the dimensions and entities are exported.

yes—Exports dimensions with breaks on witness lines so that entities are separated and the original picture is preserved.

dxf_out_stroke_text

yes, *no*

Controls the export of text as stroked out or not.

- yes—Strokes out text in DXF or DWG export. Text that is stroked out is converted to lines and dots.
- no—Does not stroke out text.

dxfio_in_sjis

yes, *no*

Determines whether SJIS is supported when importing and exporting DXF files in Japanese. SJIS is Japanese character encoding.

edge_display_quality

normal, high, very_high, low

Controls the display quality of an edge for wireframe and hidden-line removal.

- normal—Provides a normal quality of edge display.
- high—Improves the display quality by increasing tessellation by a factor of 2.
- very_high—Improves the display quality by increasing tessellation by a factor of 3.
- low—Decreases tessellation compared to the normal, speeding up the display of an object.

enable_cadra_export

yes, *no*

Enables the CADRA option in the **EXPORT** menu. This enables the creation of a CADRA-specific IGES file.

explode_iges_dimension_note

yes, *no*

Controls the display of dimensions when importing an IGES drawing file.

- yes—Explodes each IGES dimension into two entities. An independent note contains the dimension text and another dimension with an empty note.
- no—Treats dimensions as in the original file.

export_3d_force_default_naming

yes, *no*

Determines whether to use Pro/ENGINEER file names when exporting to STEP.

- yes—Uses Pro/ENGINEER file names when exporting to STEP. You must accept the default file name. Does not prompt you to specify a file name before continuing with the export.
- no—Prompts you for the name of each STEP file.

export_to_shipit

yes, *no*

Enables Ship-it interface export.

extend_cvpath

<path name>

Adds paths to the CVPATH environment variable during your Pro/ENGINEER session. The CVPATH environment variable defines the default paths to search for CADDS models.

Pro/ENGINEER searches for CADDS components in the current working directory or uses the CVPATH environment variable with the extend_cvpath configuration option as a reference for locating components in other directories.

fix_autocad_iges_text_scale

yes, *no*

Fixes scaling problems for AutoCAD releases earlier than Release 10.

yes—Corrects AutoCAD scaling problems.

fix_boundaries_on_import

yes, *no*

Controls the fixing of boundaries of imported surfaces.

- yes—Fixes boundaries by zipping gaps and correcting tangency.
- no—Does not fix boundaries.

fix_catia_iges_sym_note

yes, *no*

Corrects a problem caused by multiple rotations of a note in a CATIA IGES file.

yes—Rectifies discrepancies caused by multiple rotations of a note in a CATIA IGES file.

fix_imported_set_view_orient

yes, *no*

Sets the view characteristics for imported CATIA SET files.

- yes—Imports files with translated views.
- no—Imports files without translated views.

graphics

gl, *opengl*, starbase, xgl, x_windows

Sets the optional graphics environment used by certain platforms for running Pro/ENGINEER.

To import files in snapshot TIFF format into Pro/ENGINEER, ensure that graphics is set to x_windows on UNIX and win32_gdi on Windows.

iges_clip_view_ent

yes, no

Detemines whether IGES entities are clipped with respect to IGES views.

- yes—Clips entities outside the view outline.
- no—Does not clip entities.

iges_clip_view_note

no_clip, full_clip, partial_clip

Determines whether IGES notes are clipped with respect to the IGES views.

- no_clip—Does not clip notes.
- full_clip—Clips only notes completely outside the view outline.
- partial_clip—Clips notes that are even partially outside the view outline.

iges_export_dwg_views

yes, *no*

Determines the export of drawing-view information.

- yes—Exports drawing-view information.
- no—Does not export drawing-view information.

iges_in_106_f2_as_spline

yes, *no*

Determines how the IGES Copious Data Entity is imported into Pro/ENGINEER.

- no—Imports IGES Copious Data Entity (type 106, form 2) as a set of points in 3D space.
- yes—Imports IGES Copious Data Entity as a 3D spline.

iges_in_assoc_dim_geom_21

yes, *no*

Determines whether associative dimensions imported from IGES remain associative on import.

- yes—Processes the Associativity Instance entity (type 402, form 21) during import when the associative_dimensioning drawing setup option is set to yes.
- no—Imported dimensions do not retain associativity.

iges_in_dwg_color

yes, *no*

Determines the import of RGB information in the IGES files. The color-definition entities in the IGES files are either imported or ignored. This configuration option is available as **Import User Colors** in the **Import IGES** dialog box.

- yes—Imports RGB information. The color-definition entities in the IGES files are imported to the drawings as user-defined colors. All entities using the color-definition entities are set to use the user-defined colors.
- no—Ignores the color-definition entities in the IGES files. All entities referencing these colors are set to use the assigned color in the color-definition entity.

iges_in_dwg_line_font

yes, *no*

Controls the import of user-defined line fonts. This configuration option is available as **Import User Line Fonts** in the **Import IGES** dialog box.

- yes—Gives default names to user-defined line styles that do not have names. The names are in the order IGES_1, IGES_2, and so on.
- no—Imports the user-defined line fonts as a solid line font.

iges_in_group_to_dwg_layer

yes, *no*

Determines the conversion of an IGES group to drawing layers. This configuration option is available as **Import Groups As Layers** in the **Import IGES** dialog box.

- yes—Converts an IGES group to drawing layers.
- no—Does not convert the IGES group to drawing layers.

iges_note_disp

as_geometry, *all_views*, no_views, as_is

Determines the action to be taken when IGES note entities do not point to any views or the drawing entity.

The iges_zero_view_disp configuration option determines the action when IGES geometry does not point to any views or the drawing entity.

iges_out_all_srfs_as

default, 114, 128

Determines surface types that are exported to the IGES file format.

- default—Exports all surfaces as appropriate IGES surfaces.
- 114—Applies only to representations of surface shape. Trimmed surface entities (type 144) are exported regardless of this setting.
- 128—Exports all surfaces as IGES B-spline surfaces.

iges_out_assembly_default_mode

flat, one_level, all_levels, all_parts

Specifies the default for export of assemblies through IGES. The optional values are:

- flat—Converts the assembly to a single-level geometric model.
- one_level—Exports only the assembly structure with pointers to component files.
- all_levels—Exports the assembly structure to a single file and the components to IGES files.
- all_parts—Exports an assembly to IGES as multiple files containing geometry information about the assembly components and assembly features, if any.

iges_out_catia_gdt_width

yes, *no*

yes—Enables the required width of a gtol symbol to be exported to CATIA.

iges_out_catia_notes

yes, no

yes—Breaks a large note, more than 70 strings, into smaller notes, each producing an IGES entity.

iges_out_dwg_color

yes, *no*

Determines the export of RGB information and the user-defined colors in the drawing to the IGES file format.

- yes—Enables the export of RGB information to IGES files. The user-defined colors in the drawing are exported to an IGES file as color-definition entities. All entities using these colors have a pointer to the corresponding color-definition entity in the IGES file.
- no—Ignores the user-defined colors in the drawing when exporting IGES file. All entities using these colors are set to use the white color in the IGES file.

iges_out_dwg_line_font

yes, *no*

Controls the export of user-defined line fonts through IGES.

no—Exports all geometry as solid fonts.

iges_out_ent_as_bspline

true, *false*

Controls the export of geometric entities in drawings to IGES files.

- true—Exports geometric entities other than lines or arcs as third-degree B-splines.
- false—Does not export entities as third-degree B-splines.

iges_out_jamais_compliant

yes, *no*

- yes—Specifies IGES output with special JAMA-IS subset specification, compliant with version 1.02 of JAMA-IS (Japan Automobile Manufacturers Association IGES Subset Specification).
- no—Specifies normal IGES.

iges_out_mil_d_28000

yes, *no*

yes—Specifies the IGES output that uses the MIL-D-28000 entity subset.

iges_out_spl_crvs_as_126

yes, no

Converts part geometry spline curves to IGES entity 126 (B-spline) when creating an IGES file.

iges_out_spl_srfs_as_128

yes, no

Converts part geometry spline surfaces to IGES entity 128 (B-spline) when creating an IGES file.

iges_out_start_note

yes, *no*

yes—Exports the text specified by the system_iges_header_file and the user_iges_header_file as a note on the drawing.

Use the put_iges_drawing_entity configuration option to control the placement of the note.

iges_out_symbol_entity

yes, no

Exports a drawing symbol as an IGES general symbol entity, IGES type number 228 (yes) or as its component entities, notes, and lines.

iges_out_trim_curve_deviation

value, *current accuracy*

Sets the maximum value for the distance between an XYZ trimming boundary curve and the underlying surface of a trimmed surface.

iges_out_trim_xyz

yes, no

Determines whether XYZ data is exported in addition to UV data for trimmed surfaces.

iges_out_trm_srfs_as_143

yes, *no*

yes—Exports surfaces to IGES entities 141 and 143. This overrides the iges_out_trm_xyz option.

iges_zero_view_disp

all_views, no_views, as_is

Determines the action when IGES geometry does not point to any views or the drawing entity.

- all_views—Creates a copy of the entity for each view using view transformation.
- no_views—Does not create the entity.
- as_is—Creates the entity only once, using its own view transformation information.

interface_quality

0, 1, 2, *3*

Determines the amount of work performed when checking for overlapping lines in a plot or a 2D export file, such as IGES.

- 0—Dumps lines in the interface without checking for overlapping lines or collecting lines of the same pen color.
- 1—Does not check for overlapping lines, but collects lines of the same pen color for plotting.
- 2—Partially checks edges with two vertices and collects lines of the same pen color for plotting.
- 3—Completely checks all the edges against each other, regardless of the number of vertices, fonts, or colors. Lines of the same pen color are collected for plotting.

intf_cadds_version

12 , 13

Controls the version of a CADDS 5 file when exporting a Pro/ENGINEER part or assembly to CADDS 5. Allows Pro/ENGINEER to switch between the default and the alternate CADDS 5 converter version for the interface with CADDS 5.

Set this option at initial startup or runtime using **Tools** > **Options**.

intf_collapse_geom

yes, no

yes—Collapses the Pro/ENGINEER parametric surface into the import feature.

intf_in_arclength_reparam

yes, *no*

- yes—During import, the system attempts to reparameterize nonanalytical surfaces whose parameterization differs significantly from arc length. The nonanalytical surfaces that are reparameterized are:
  - Tabulated Cylinders
  - Surfaces of Revolution

    - Ruled Surfaces
    - Spline Surfaces
    - B-spline Surfaces
- no—No attempt is made to apply this arc length reparameterization.

intf_in_blanked_entities

yes, no

Filters the import of entities based on their blank status in the import file.

- yes—Imports the entity to Pro/ENGINEER and places it on a layer called INTF_BLANK.
- no—Does not import blanked entities.

intf_in_dwg_pnt_ent

yes, *no*

Converts an IGES or DXF point entity to a drawing point. This configuration option is available as **Import Points** in the **Import DXF**, **Import DWG**, and **Import IGES** dialog boxes.

intf_in_dwg_view

3D_views, *2D_views*, no

Determines whether associativity within an IGES view is preserved when an IGES drawing is imported to Pro/ENGINEER. This configuration option is available as **VIEWS** in the **Import IGES** dialog box.

- 3D_views—Creates 3D views when a 3D model exists in the file.
- 2D_views—Imported IGES views work as 2D views.
- no—Imported IGES views are exploded and become unrelated.

intf_in_extract_profiles

none, comp, all

Controls the conversion of planar composite curves to datum curves for feature creation. This is when you import an IGES, STEP, or CATIA file containing planar composite curves into a Pro/ENGINEER part or assembly.

- none—Does not extract profile curves during import.
- comp—Imports only planar composite curves.
- all—Imports all planar curves.

intf_in_granite_direct_enable

yes, no

Allows you to choose the method of opening Granite-based file formats in Pro/ENGINEER. The method of directly opening the Granite-based file formats as Pro/ENGINEER models is the default. The direct open method preserves the feature-structure of the Granite file in the resultant Pro/ENGINEER file though these features are 'read-only' in Pro/ENGINEER.

- yes—Directly opens a Granite-based file in Pro/ENGINEER as a Pro/ENGINEER model with read-only features or components, or both.
- no—Imports a Granite-based file into Pro/ENGINEER using the standard import method, that is, as import features or components, or both. You can edit the import features or components.

intf_in_layer_asm_dialog

yes, *no*

- yes—Opens a dialog box to control the import of both layers and assemblies.
- no—Does not control the import of layers and assemblies.

intf_in_surf_boundary_pref

uv, xyz

Determines which data set to use during the import. In all interfaces that handle a 3D surface, trimming of the edges of faces are represented redundantly using both UV and XYZ edges.

intf_in_treat_polyline_as

single_polyline, single_spline, set_of_curves

Enables you to choose the representation of imported independent XYZ polylines. The system interprets the imported polylines based on the option selected and displays them accordingly.

- single_polyline—Displays the polyline as a curve that can be selected as a single polyline.
- single_spline—Displays the polyline as a single smooth curve.
- set_of_curves—Displays the polyline as a set of smooth curves connected by lines and splines, depending on the entity imported.

intf_in_use_template_models

yes, *no*

Determines the use of start part and start assembly template files when importing data. Templates are not applied to data that is appended to existing models.

Use `template_solidpart` and `template_designasm` configuration options to locate these templates.

`yes`—Pro/ENGINEER imports the new part or assembly from other file formats into the default start part and start assembly template files.

intf_out_as_bezier

Exports B-splines as Bezier surfaces.

intf_out_assign_names

`no_name`, `user_name`, `id_name`

Determines how entity names are handled when an object is exported to the STEP format from the Part or Assembly mode. You can assign a name to an entity in a STEP file before export. You can give unique names to Pro/ENGINEER datum points, datum axes, datum curves, surfaces, edges, and quilts.

The receiving system can use this information to identify entities or reference these names in other programs that read STEP data. The entity name propagates to all entities used to define the top-level entity name in Pro/ENGINEER. For example, a surface named `topsurf` would also have boundary curves named `topsurf`.

- `no_name`—Does not export entity names, a null value appears.
- `user_name`—You can assign names for the STEP entities. Use **Setup** > **Name** to name the entities. These names override the ID names.
- `id_name`—Assigns unique names consisting of the Pro/ENGINEER internal ID number to STEP entities.

intf_out_auto_layer_ids

`yes`, *`no`*

- `yes`—Automatically assigns interface IDs for layers not assigned IDs during export.
- `no`—Does not assign interface IDs for layers not assigned IDs during export.

intf_out_blanked_entities

`yes`, `no`

Controls the export of blanked layers. Filters export of entities based on their `blank` status.

- `yes`—Exports entities on blanked layers.
- `no`—Does not export entities on blanked layers. The entities on blanked layers are ignored.

intf_out_cat_start_model

`path to the CATIA .model start parts file`

Controls the use of the attributes or parameters defined in the CATIA `.model` custom or predefined start parts file when you export a Pro/ENGINEER part or assembly to CATIA.

- When you specify the path to the CATIA `.model` start parts file, Pro/ENGINEER parts and assemblies use the model parameter values defined in the CATIA `.model` start parts template files.
- When you do not specify the path to the CATIA `.model` start parts file, Pro/ENGINEER parts and assemblies use the default model parameter values.

intf_out_layer

`block_layer`, `part_layer`, `none`

- `block_layer`—Exports components (parts), symbols, groups (for each view), and tables of Pro/ENGINEER drawings as AutoCAD blocks.
- `part_layer`—Exports data from Pro/ENGINEER drawings to DXF and DWG format.
- `none`—Exports Pro/ENGINEER drawings in DXF and DWG format without creating blocks or mapping layers and line styles.

intf_out_layer_rename_table

Enables you to assign interface IDs to layers during export.

intf_out_max_bspl_degree

`integer (range 3 - 16)`

Controls the maximum degree of the exported B-spline surfaces when exporting through the IGES format.

Planar surfaces are not controlled by any configuration option and are exported to IGES as B-spline surfaces of degree 1 by 1.

intf_out_text_length

`as_is`, *`full_size`*, `adjusted`

Sets the text length in the exported 2D file.

- `as_is`—The width of each character is the width of the strokes.
- `full_size`—The width of each character is the width of the character text box.
- `adjusted`—Adjusts spacing so that no extra space is left and the end or start character does not overlap.

intf_use_variable_size

yes, no

Controls the size of the drawing sheet, depending on whether the imported drawing sheet size varies from the standard Pro/ENGINEER drawing sheet size and the drawing sheet size information contained in the imported files of the IGES, STEP, DXF, DWG, MEDUSA, STHENO/PRO, CADAM, and CGM formats.

For a DXF drawing file with multiple drawing sheets of variable size that is imported as a single drawing with multiple sheets, the variable sizes of the imported drawing sheets are maintained in Pro/ENGINEER.

- yes—Creates drawing sheets of different but appropriate sizes when the drawing sheet sizes vary from the standard Pro/ENGINEER drawing sheet size. The variable drawing sheet sizes are created depending on the drawing sheet size information specified in the imported files.
- no—Places the imported drawing on the standard-size drawing sheet in Pro/ENGINEER. For a DXF file with multiple drawing sheets of variable sizes, for each sheet of the drawing, a standard-size drawing sheet that is nearest in size to the original drawing sheet is created in Pro/ENGINEER.

This configuration option is available as **Create Variable Sheet Size** in the **Import DXF**, **Import DWG**, and **Import IGES** dialog boxes.

intf2d_fit_incompatible_data

yes, *no*

Controls the compatibility between 2D external formats and Pro/ENGINEER.

yes—Fixes compatibility problems between 2D external formats, such as IGES and DXF, and Pro/ENGINEER.

intf2d_iges_out_hatch

yes, *no*

Determines how cross hatches in 2D IGES files are exported. Cross hatches are exported as separate single geometric entities or as IGES Element Type 230.

- yes—Exports draft cross hatches as IGES Element 230. That is, a sectioned area is created for each draft cross hatch.
- no—Exports draft cross hatches as separate single geometric entities or independent lines.

intf2d_in_acad_ignore_3d

yes, *no*

Allows you to skip the 3D solid entities in the 3D DXF or DWG files imported into Pro/ENGINEER in the Drawing mode.

- yes—Ignores the 3D solid entities in the 3D DXF or DWG files imported into Pro/ENGINEER in the Drawing mode and processes the 2D draft entities.
- no—Processes the 3D solid entities along with the 2D entities in the 3D DXF or DWG files imported into Pro/ENGINEER in the Drawing mode and creates assemblies with the 3D data.

This configuration option is available as the **Import 3DSOLID entities** option on the **Import DXF** and **Import DWG** dialog boxes.

intf2d_in_create_multiline_note

yes, no

Controls the conversion of multiple-line text for all 2D file formats, except SET and CADAM, to a single multiple-line note or multiple single-line notes. If each line of the text in the imported file is of a different font, you can preserve the original fonts in the multiple-line notes.

- yes—Creates a single multiple-line note with the default style settings. Style settings of the notes are ignored during import.
- no—Creates multiple single-line notes with the original fonts and styles.

This configuration option is available as the **Create multi-line text** option in the **Import DXF**, **Import DWG**, and **Import IGES** dialog boxes.

intf2d_in_iges_hatch_bnd_layer

yes, *no*

Determines whether to place the boundary lines of sectioned areas on a single layer when hatches are included in the IGES file that you are importing to Pro/ENGINEER.

You can place the boundary lines of the hatches on a layer in Pro/ENGINEER and change the import layer display status to Blank so that the boundary lines are not visible after the import of the IGES file.

- yes—Places boundary lines of sectioned areas from the IGES file on a layer named IGES_HATCH_BOUNDARY.
- no—Does not place boundary lines of sectioned areas on a layer.

intf2d_out_acad_mtext

yes, no

Controls the export of multiple-line notes to the DXF and DWG file formats as a single MTEXT entity or as multiple TEXT entities.

- yes—Converts the multiple-line text notes in the Pro/ENGINEER drawing to a single MTEXT entity in the DXF or DWG file.
- no—Converts the multiple-line text notes in the Pro/ENGINEER drawing to multiple text entities in the DXF or DWG file.

This configuration option is available as the **Export multi-line note as MTEXT** option in the **Export Environment for DXF** and the **Export Environment for DWG** dialog boxes.

intf2d_out_acad_text_align

as_is, fit

Controls the alignment of text exported to the DXF and DWG file formats. You can export text as is or with the FIT alignment.

- as_is—Exports text in notes as is. The original alignment of text in the notes is preserved.
- fit—Exports a note with text stretched or squeezed to fill or fit the space between the start and end points of the note.

This configuration option is available as the **Text Alignment** option in the **Export Environment for DXF** and the **Export Environment for DWG** dialog boxes.

intf2d_out_cgm_ver

1, 3

Determines the CGM version for export.

3—Exports all curves, except lines and arcs, as Non-Uniform B-splines or Non-Uniform Rational B-splines (NURBS). Notes are exported as text entities.

intf2d_out_enhanced_ents

spline_and_hatch, spline, hatch, none

Controls the export of polylines, lines, and arcs, or hatch and spline entities to a DWG or DXF file.

- spline_and_hatch—Exports splines and hatches.
- spline_only—Exports splines.
- hatch_only—Exports hatches.
- none—Does not export any enhanced entity.

intf2d_out_open_log_window

yes, *no*

Controls the display of the log file created during the export of drawing data using the DXF, DWG, or the IGES file format. The log file contains details such as the file version, the configuration option settings, the assembly structure representation as blocks or layers, text as is or as geometric entities, and so on.

yes—You can view and verify the information, edit, and save the log file in an Information window.

intf2d_out_pnt_ent

no, yes

Exports points of a drawing in the DXF, DWG, or the IGES file format as is or as shapes.

intf3d_ideas_import_filter

yes, *no*

Controls the display of the **Layer Import Options** dialog box during the import of the I-DEAS .mf1 and .pkg files. Use the **Layer Import Options** dialog box to filter or selectively import the part and assembly models contained in these files.

- yes—Opens the **Layer Import Options** dialog box with which you can selectively import the parts and assemblies from the .mf1 and .pkg files.
- no—Does not open the **Layer Import Options** dialog box during the import of the .mf1 and .pkg files.

intf3d_ideas_install_dir

path to the I-DEAS loadpoint

Points to the I-DEAS installation. Specify the full path to the installation.

Note: If this configuration option does not point to the location of the I-DEAS installation, a dialog box with a warning prompts you to define the full path to the I-DEAS installation when you select the .mf1 or the .pkg file type for import in the **File** > **Open** dialog box.

intf3d_ideas_run_command

ideas

Specifies the startup command for I-DEAS. This configuration option is set to the value, ideas, by default.

If the I-DEAS startup command, as specified by this configuration option, is not found in the system path, an error appears in the Pro/ENGINEER message window stating that the I-DEAS startup command is not in the system path. The message prompts you to use this configuration option to set the startup command as I-DEAS.

intf3d_in_close_open_boundaries

yes, *no*

Controls fixing of open boundaries on imported surfaces. Attempts to close open boundaries of surface patches when trimming of boundaries during import fails.

- yes—Closes open trimming boundaries of imported surfaces by connecting the end-points of existing boundaries.
- no—Leaves boundaries untrimmed.

intf3d_in_enable_layer_join

yes, no

Determines how surface geometry is joined during import.

- no—Joins surfaces as the geometry is loaded from the file. No layering information is captured in the imported feature.
- yes—Joins surfaces by layer. Groups imported surfaces according to originally imported layers. Pro/ENGINEER first joins surfaces within groups corresponding to the imported layers. The rest of the geometry is joined later. This kind of joining reduces topological ambiguity in imported data.

intf3d_in_include_items

srfs, crvs, pnts, crv_pnts, srfs_crvs, srfs_pnts, *srfs_crvs_pnts*

Specifies what entity types you want to import from a file.

- srfs—Imports only surfaces.
- crvs—Imports only curves.
- pnts—Imports only points.
- crvs_pnts—Imports curves and points.
- srfs_crvs—Imports surfaces and curves.
- srfs_pnts—Imports surfaces and points.
- srfs_crvs_pnts—Imports surfaces, curves, and points.

intf3d_out_cat2_ident_crv

-1.000000

Corresponds to the tolerance curve parameter of the Pro/ENGINEER model. Set this configuration option before you export the Pro/ENGINEER model to CATIA.

The tolerance value that you specify for this option is based on the correlation between a standard Model Size and the parameter. Set any value or set the recommended value that is based on a standard Model Size of 10000.

intf3d_out_cat2_ident_pt

-1.000000

Corresponds to the tolerance point parameter of the Pro/ENGINEER model. Set this configuration option before you export the Pro/ENGINEER model to CATIA.

Base the tolerance value on the correlation between a standard Model Size and the parameter. Set any value or set the recommended value that is based on a standard Model Size of 10000.

intf3d_out_cat2_infinity

-1.000000

Corresponds to the tolerance line parameter of the Pro/ENGINEER model. Set this configuration option before you export the Pro/ENGINEER model to CATIA.

The tolerance value that you specify for this option is based on the correlation between a standard Model Size and the parameter. Set any value or set the recommended value that is based on a standard Model Size of 10000.

intf3d_out_cat2_model_sz

-1.000000

Corresponds to the model size model parameter of a Pro/ENGINEER part before you export the part to CATIA. Set this configuration option before you export the Pro/ENGINEER model to CATIA to ensure the accuracy of the model size in the exported part.

The tolerance values that you specify correspond to the model parameters and are based on the correlation between a standard Model Size and the parameters. Set any value or the recommended value that is based on a standard Model Size. The default values for the parameter is based on the standard Model Size of 10000.

intf3d_out_cat2_sag

-1.000000

Corresponds to the tolerance sag parameter of the Pro/ENGINEER model. Set this configuration option before you export the Pro/ENGINEER model to CATIA.

The tolerance value that you specify for this option is based on the correlation between a standard Model Size and the parameter. Set any value or the recommended value that is based on a standard Model Size of 10000.

intf3d_out_cat2_step

-1.000000

Corresponds to the tolerance distance parameter of the Pro/ENGINEER model. Set this configuration option before you export the Pro/ENGINEER model to CATIA.

The tolerance value that you specify for this option is based on the correlation between a standard Model Size and the parameter. Set any value or the recommended value that is based on a standard Model Size of 10000.

intf3d_out_datums_by_default

yes, *no*

Determines whether datum curves are included when exporting IGES files in Batch mode.

- yes—Includes datum curve information.
- no—Does not include datum curve information.

intf3d_out_default_option

wireframe, *surfaces*, wireframe_surfaces

Determines the type of information included when exporting IGES files in Batch mode.

- wireframe—Wireframe only.
- surfaces—Surface quilts only.
- wireframe_surfaces—Wireframe and surface quilts.

intf3d_out_extend_surface

yes, no

Specifies how surfaces are handled for exporting files to other systems.

- yes—Extends surfaces until they intersect.
- no—Exports surfaces as they are, with no extension.

intf3d_out_force_surf_normals

yes, *no*

Controls normals of nonanalytic surfaces.

VDA always forces surface normals. Although this option facilitates data exchange with CATIA, it may also facilitate other receiving CAD systems.

- yes—Forces the surface normals to point in a consistent direction.
- no—Does not force the surface normals to point in a consistent direction.

intf3d_out_surface_deviation

-1.000000

Enables you to set a maximum allowed deviation between the original and the resulting surfaces during the conversion of Pro/ENGINEER surfaces to spline surfaces. For example, for IGES this conversion occurs only when the iges_out_all_srfs_as option is set to 114. For CATIA and VDA, this conversion always occurs.

Setting the maximum deviation enables you to convert surfaces with a better approximation than by using the current accuracy of the model.

Specify a positive value in the current model units to set the deviation. If the value that you specify is outside the accepted range, the system uses the default model accuracy.

intf3d_parasolid_export_schema

SCH_10004

Allows you to set the export schema when exporting Pro/ENGINEER parts and assemblies to the Parasolid format. The values for the export schema range between SCH_3000 and SCH_13006.

Set the value of intf3d_parasolid_export_schema to any of the following schema:

- SCH_13006
- SCH_13005
- SCH_12103
- SCH_12102
- SCH_12006
- SCH_11004
- SCH_11003
- SCH_10100
- SCH_10004
- SCH_10002
- SCH_9100
- SCH_9008
- SCH_9003
- SCH_9001
- SCH_8101
- SCH_8008
- SCH_8005
- SCH_8002
- SCH_8000
- SCH_7016
- SCH_7015
- SCH_7014
- SCH_7007
- SCH_7002
- SCH_6021
- SCH_6020

- SCH_5059
- SCH_5058
- SCH_5057
- SCH_5056
- SCH_5054
- SCH_5053
- SCH_5051
- SCH_5050
- SCH_5049
- SCH_5041
- SCH_5033
- SCH_5032
- SCH_5031
- SCH_5030
- SCH_5021
- SCH_5015
- SCH_4039
- SCH_4035
- SCH_4032
- SCH_4031
- SCH_4030
- SCH_4022
- SCH_4011
- SCH_3000
- SCH_210_1012

intf3d_ug_install_dir

path to Unigraphics loadpoint

Specifies the path to the Unigraphics loadpoint directory. Specify the absolute path.

If you have not set this configuration option, then when you attempt to import or export a Unigraphics .prt file with the **Unigraphics File (*.prt)** file type set, a warning in the Pro/ENGINEER window prompts you to use this configuration option to define the location of the Unigraphics installation.

medusa_2d_config_file

Specifies the MEDUSA 2D Interface configuration file. Use the absolute or relative path to the file.

pdf_plot_config

Set the Portable Compiled Format (PCF) filename to be used when saving to PDF. For example, `pdf_A4.pcf`.

The PCF file uses the Adobe Distiller settings such as plot resolution, color, fonts and various other settings defined in the Adobe Distiller user interface, to generate the PDF.

Pro/ENGINEER looks for the PCF file, first in the working directory, and then in the directory specified by the `pro_plot_config_dir` configuration option.

If the configuration option is not set, or if Pro/ENGINEER cannot find the PCF file specified by the `pdf_plot_config` configuration option, or the plotter value in the PCF file is not set, then the default `POSTSCRIPT` setting for PDF export is used.

pen_slew

Sets the pen speed for the x- and y- directions for plotters that are compatible with this option.

plot_file_dir

`<directory name>`

Specifies the directory to which plot files are written. Use the full path to avoid problems. For example, `/home/users/plotfiles`.

plotter

`<desired plotter name>`

Establishes the default plotter if the plotter name is specified when making plot files.

plotter_command

`windows_print_manager, print /d:\\print_server_name\printer_share_name, copy %1 \\printer_server_name\printer_share_name`

Sets the command that you are using to start a plot on your system. For Windows, the `windows_print_manager` option configures Pro/ENGINEER to plot to a recognized device in the Windows print manager.

The `windows_print_manager` plotter command, used to plot directly from Pro/ENGINEER on Windows, does not work in Pro/BATCH.

pro_gplug_dir

<directory name>

Specifies the directory in which Granite application plugins are located. Use the full path to avoid problems.

pro_stheno_command

Points to the STHENO/PRO application startup command, sthenolaunch.exe, before starting STHENO/PRO from within Pro/ENGINEER.

Note: Exit Pro/ENGINEER if the pro_stheno_command configuration option is not set or the STHENO/PRO user interface elements are not visible or accessible. Set the pro_stheno_command configuration option so that it points to the STHENO/PRO application startup command and start Pro/ENGINEER again.

pro_plot_config_dir

<directory name>

Sets the directory of your user-defined plotter configuration file. Use the full path to avoid problems. For example, /home/users/plot_dir.

put_iges_drawing_entity

yes, no

Specifies whether or not to suppress the output of the IGES drawing entity, #404, to the IGES file.

no—The drawing entity is not output.

read_vda_in_pset_as_spline

yes, no

Specifies whether a VDA Pset entity is being imported to Pro/ENGINEER as a spline and controls the data point sets imported from a VDA file.

- yes—Imports data points as spline curves.
- no—Does not import data point sets as spline curves.

recompute_iges_dim_value

yes, *no*

Determines whether to recompute associative dimensions on import of an IGES file.

- yes—Instead of the IGES dimension text, the actual recomputed dimension value appears with the imported dimension.
- no—The IGES dimension text appears with the imported dimensions.

search_path

directory paths

Specifies a list of directories to search, in the order listed, for object or file retrieval. These directories, along with the current (working) directory and any directories specified in the search.pro file make up Pro/ENGINEER's search path.

The directory paths can be relative or absolute. You can use special characters, such as ".." in UNIX and Windows, in specifying a relative path name.

Relative path names are initially resolved relative to the startup directory. If you subsequently reload the configuration file, the system reevaluates the paths relative to the current (working) directory and appends the new directories, if any, to the search path (the previous path remains in place). Therefore, specify the full paths to avoid problems if you change the working directory or use the same configuration file in another startup directory.

The option can have several paths on a single line separated by commas, semicolons, or spaces. Use consistently the delimiter you choose. The option can appear any number of times in the configuration file. It is, therefore, not necessary to have more than one path to a line. If objects with the same name are stored in more than one search path, the system retrieves the first one that it finds, regardless of the object that is the most recent.

Search paths can also include previously-defined environment variables. Begin the variable with $ in the search-path definition. For example, the environment variable OBJ_TYPE is used as follows:

```
search_path /partlib/$OBJ_TYPE/objs
```

When defining a search path in the Windows operating system, precede the backslash with another formatting character.

shade_surface_feat

yes, no

Controls the shading of surface features.

- yes—Shades surfaces.
- no—Does not display surface features with shading.

step_appearance_layer_groups

no, yes

Enables the generation of appearances, layers, and groups for STEP AP214 and STEP AP20.

- no—Enables the generation of appearances, layers, and groups for STEP standard AP214.
- yes—Enables the generation of appearances, layers, and groups for STEP standard AP20. Maintains layer export and import consistency between AP203 and AP214.

step_export_ap214_asm_def_mode

single_file, separate_parts_only, separate_all_objects

Determines the file format of Pro/ENGINEER assemblies exported to STEP. All options generate an assembly-level STEP file.

step_export_dwg_views

as_3d_views, as_3d_views_assoc_draft, *no*

Determines how 3D model geometry is to be exported to a DWG file.

- as_3d_views—Exports 3D model geometry with its associative views.
- as_3d_views_assoc_draft—Exports 3D model geometry with its associative views and view-related annotations.
- no—Exports only 2D representations of the 3D models.

step_export_format

ap203_is (the default in 3D mode), ap214_cd (the default in drawing mode), ap202_is, 203_is_ext, ap214_dis, ap209_dis, ap214_is

Determines the format when you export 3D model and drawing data to STEP.

- ap203_is—Exports a 3D model using the ISO 10303 AP203IS STEP application protocol and conformance class.
- ap214_cd—Exports the drawing using the AP214CD2 STEP application protocol and conformance class. Formats the output with geometry that meets the specification for the schema for AP214 cc1.
- ap202_is—Exports the drawing using the AP202IS STEP application protocol and conformance class.
- 203_is_ext—Exported file includes AP203 validation properties by default. To include AP203 extensions by default in the file that you are exporting to STEP, set the step_export_format configuration option to 203_is_ext. Setting step_export_format to 203_is_ext, exports data to a STEP file that conforms to the International Standard of STEP with the following extensions:
  - cla—Colors and layers
  - gvp—Geometric Validation
  - ast—Associative Text
- ap214_dis—Exports the drawing using the AP214DIS STEP application protocol and conformance class. Validation properties are also exported.
- ap209_dis—Exports the 3D model using the AP209DIS STEP application protocol and conformance. Edges, boundary conditions, constraints, loads, mesh, and mid planes data are not supported for export.

- ap214_is—Exports the 3D model using the AP214IS STEP application protocol and conformance class.

step_in_style_bndry_as_fill_area

yes, *no*

Provides colors support for importing Unigraphics STEP files. Valid only when importing STEP files.

yes—Provides colors support for importing Unigraphics STEP files.

system_iges_header_file

<filename>

Inserts a specified text file into the Start section of the IGES files. When used with user_iges_header_file, the system text appears first.

tablet_device_name

device name, calcomp

Specifies the name of a digitizing tablet.

tiff_compression

none, G4

Determines whether or not to export to TIFF as compressed files using G4 compression.

tiff_type

palette, *rgb*, grayscale, mono

Determines the type of TIFF items that are exported.

topobus_enable

yes, *no*

yes—Enables Associative Topology Bus (ATB) functionality.

try_g2_fix_on_import

yes, no

Cleans and smooths noise in B-Spline surfaces before converting the B-Spline surfaces to Pro/ENGINEER surfaces.

use_cadam_plot_data

yes, *no*

Determines whether the information in the Plot Axis System element must be taken into account when importing a CADAM drawing.

- yes—If the drawing being imported contains the Plot Axis System element, then the system asks you to define the format based on the Plot Axis System. If you answer Y, the system selects the format with respect to the height, width and origin, as defined by the Plot Axis System. If you answer N, the system determines the format in the same way that it does for regular CADAM files.
- no—The system ignores the Plot Axis System element.

use_export_2d_dialog

yes, no

Opens or does not open the **Export Environment for DXF**, the **Export Environment for DWG**, or the **Export Environment for IGES** dialog box.

yes—The **Export Environment for DXF**, the **Export Environment for DWG**, or the **Export Environment for IGES** dialog box with options specific to the respective file types opens.

use_iges_font_1003

yes, no

Disables the use of IGES font 1003.

no—Font sets 1001 and 1002 are used first.

use_iges_kanji_font_2001

yes, *no*

Determines whether to convert Kanji notes in Pro/ENGINEER to IGES Kanji notes (font code) when exporting.

- yes—Converts Kanji notes to IGES Kanji notes.
- no—Converts using Font 1.

When importing, Kanji notes are converted to Pro/ENGINEER Kanji regardless of the value for this option.

user_iges_header_file

<filename>

Inserts the specified text file in the Start section of the IGES file. You can use valid parametric note symbols to be replaced during the export in this file. For example, in the text file you can use &dwg_name which is replaced by the actual drawing name when a drawing is exported. Similarly, &model_name is replaced with the part or assembly name when exporting from these modes.

Use the full path to avoid problems, for example, /home/users/iges_header_file.

vda_header

<filename>

Specifies the complete file name of the text file containing VDA header information. If you intend to use the same header for all VDA files, specify the full pathname, for example, /home/users/library/vda_head.txt.

vrml_anchor_url

name, full_name, base_name, proe_name, default

Enables you to place an anchor on a specified VRML component during the export to VRML. The keywords are optional.

- name—Uses the VRML file name without the extension.
- full_name—Uses the VRML file name with the extension.
- base_name—Uses the Pro/ENGINEER object name without the extension.
- proe_name—Uses the Pro/ENGINEER object name with the extension.
- default—Produces an empty WWW anchor for VRML export or produces the database as it was done for VRML export in Pro/PROCESS.

vrml_background_color

yes, *no*

Controls the export of a model to the VRML format with or without Pro/ENGINEER background color.

- yes—Exports a model to VRML with Pro/ENGINEER background color.
- no—Does not export a model to VRML with Pro/ENGINEER background color.

vrml_explode_lines

yes, no

Exports a model to the VRML format with or without the explode lines.

- yes—Exports a model to VRML with explode lines with assembly or assembly process data.
- no—Does not export a model to VRML with explode lines with assembly or assembly process data.

vrml_export_resolution

high, *medium*, low

Controls the number of levels of detail (LODs) in models exported to the VRML format.

- high—Up to 10 levels of detail. The highest level, Level 10, has the same resolution as a Pro/ENGINEER model.
- medium—Up to 9 levels of detail. The highest level, Level 9, is one level below that of the Pro/ENGINEER model.
- low—Up to 5 levels of detail. The highest possible level, Level 8, is two levels below that of the Pro/ENGINEER model.

Note: The level of detail (LOD) in a Pro/ENGINEER model is Level 10.

vrml_export_version

1.0, *2.0*

Controls the VRML version for export.

vrml_file_duplicate_material

yes, *no*

Controls the retention of component colors.

- yes—Ensures that the model components retain their true color.
- no—In some viewers, component colors may not be consistent.

vrml_multiple_views

all, none, top

Determines the views that are exported to the VRML file.

- all—Exports all view names to the VRML database.
- top—Exports only the top assembly view names to the VRML database.
- none—Does not export views to the VRML database.

vrml_parameters

designated, all, none

Determines the parameters exported for the part.

- designated—Exports designated parameters and dimensions.
- all—Exports all user-parameters.
- none—Does not export user-parameter data.

vrml_simprep_export

yes, *no*

Determines if the in-session top assembly simplified representations are exported to the package files.

- yes—Specifies direct and in-memory export of the top assembly simplified representation to the Pro/FLY-THROUGH package (.pkg) files.
- no—Does not export the simplified representations.

wf_keep_analyt_srf

yes, no

Preserves analytical surfaces, such as flat surfaces, cylinders, and cones, by an automatic cleanup operation and creates their edges by intersection.

www_add_aux_frame

yes, *no*

Customizes the output so that you have an additional frame for each step of the process or for an assembly publication.

- yes—Pro/ENGINEER creates an auxiliary aux.html file for the assembly process-in each step00 directory to be replaced by your own <filename>.html file
- no—Does not create the auxiliary file.

www_export_geometry_as

cgm_vrml, all, *jpg_vrml*, vrml, cgm, jpg

Specifies the output format to one of the values:

- cgm_vrml—Formats ouput data to CGM and VRML.
- all—Formats output data to CGM, JPG, and VRML.
- jpg_vrml—Formats output data to JPG and VRML.
- vrml—Formats output data to VRML only.
- cgm—Formats output data to CGM and VRML. Sets up the site to show only CGM.
- jpg—Formats output data to JPG and VRML. Sets up the site to show only JPG.

www_tree_location

out, in

Specifies the location of the model tree in the browser window.

- out—Opens the model tree in a separate window.
- in—Includes the model tree in the Web page and removes the **Tree** checkbox from the control panel.

Working with Imported Geometry

Basic Methods

About Working with Imported Geometry

Working with imported data depends on the number of changes to the geometry expected, the type of features to be added, and information about the geometry required for the design.

Imported data consists of 2D or 3D wireframe, 3D surface, or a combination of data types. Importing a geometry file from another CAD system adds an imported feature to a Pro/ENGINEER model.

After you incorporate a feature into the Pro/ENGINEER model, you can:

- Add other features.
- Use reference dimensions to annotate the imported feature.
- Create and dimension a drawing of the imported geometry.
- Make a solid Pro/ENGINEER feature from the imported geometry depending on the type of imported geometry and the way it is to be used.

Examples: Methods of Working with Imported Geometry

Methods of working with imported geometry follow:

Use the Geometry as Imported

You can use the imported geometry as is with standard parts that do not change because:

- They come from a standard library, for example, bolts or brackets from a library of existing parts.
- The part is the responsibility of another department or company, for example, a style surface from the Industrial Design department.

You can:

- Use the imported geometry as the basis for additional features.
- Replace geometry with updated versions.
- Place the imported geometry in assemblies.
- Reference the imported geometry for component creation or placement, or both, component creation and placement.
- Place the imported geometry in a drawing.

If you must modify geometry extensively or often, you can modify the imported entities nonparametrically or parametrically to recreate the entire model.

Modify Geometry Nonparametrically

Changes such as altering the bend angle of a wireframe bracket or moving a snap mount on a surface model are straightforward. If the geometry does not require frequent updating, parametric behavior may not be necessary. You can use the geometry as imported or as is with standard parts.

Recreate Entire Model Parametrically

You can recreate the entire part. This way, the entire part is parametric, with the advantages of associativity capturing design intent, and feature-based design.

Create or Recreate Important Geometry Only

If you have to modify certain portions of the geometry often, you can recreate only those portions or create solid features based on only some of the geometry. For example, a Legacy part used as a housing for electronics may need to attach to many different versions of the product. Recreating the geometry of the snap mounts only provides parametric functionality, like placement based on product version and family table functionality, for only the important geometry that changes often.

About Wireframe Data

When only wireframe data is available, or when data is inaccurate so that surface information cannot be imported, you can construct a solid model of the entire part or create solid features based on important geometry only, using the wireframe as a reference.

To create solid geometry from a wireframe, you can:

- Create datum planes through the wireframe entities and use the entities as references when creating the features.

  You may choose not to create solid geometry for the entire imported part.

- Create datum curves on top of wireframe entities and use the datum curves as boundaries for creating surfaces.

You can also export the wireframe associated with a solid model, including any visible surface quilts, into a single file. You can place a wireframe on a layer and blank it, if required.

If the imported wireframe geometry contains gaps between entities, use Pro/LEGACY to repair the problems.

To Create a Solid Model from a Wireframe

1. Click **Insert** > **Model Datum** > **Plane** to create datum planes for sketching references through the wireframe entities. The **DATUM PLANE** menu appears.
2. Click **Through**. Select references.
3. Click **GEOM TOOLS** > **Use Edge** to construct solid features by sketching.

Note: To remove parent-child dependency between Pro/ENGINEER features and the imported feature, remove the alignment of geometric entities created using **Use Edge**. Dimension the entities relative to the default datums and to each other.

4. Place the wireframe on a layer and blank the layer.

About Surface Data

It is best to import solid models from second generation CAD systems into Pro/ENGINEER as surfaces.

You can construct a solid from the imported surface data in several ways depending on the information available in the imported feature. A solid can be created automatically out of the imported data during the import process if Pro/ENGINEER has sufficient data, for example, closed quilts. The system tries to create a solid protrusion if the imported data contains closed quilts. The **Choose Solid Options and Placement** dialog box opens when closed quilts are imported into existing solid part geometry.

If the imported geometry contains surfaces that are not connected, the imported feature appears as a collection of quilts or a quilt with gaps. After import, convert the imported feature into a solid.

To Generate a Solid Using Surface Data

When all surfaces in the imported data are not joined automatically,

1. Select the feature in the Model Tree.
2. Right-click. A shortcut menu appears.
3. Click **Edit Definition** on the shortcut menu.
4. Click **Geometry** > **Heal Geometry** > **Manual** to redefine the imported feature and rejoin the quilts.
5. If you cannot join the remaining surface edges to create a closed volume, use one of the strategies for working with problem geometry.
6. After defining a single closed volume, click **Edit** > **Solidify** to create the solid feature.
7. If the imported feature is displayed as an open quilt that cannot enclose a single volume, click **Edit** > **Solidify** to create a thin solid.

To Assure the Accuracy of Imported Data

1. Set the configuration options according to your requirements.
2. Import the file with the default accuracy to determine the appropriate absolute accuracy.
3. Import the file again, specifying the absolute accuracy value that best suits the geometry.

Configuration Options for Imported Geometry

When you import and export geometry, you must specify in the `config.pro` file any configuration options affecting the transfer, and load the `config.pro` file into Pro/ENGINEER before transfer to ensure appropriate conversion behavior.

Before you import geometry from another CAD system, set the following configuration options:

- `intf3d_in_close_open_boundaries`
- `try_g2_fix_on_import`
- `intf_collapse_geom`
- `wf_keep_anlyt_srf`
- `intf3d_in_enable_layer_join`
- `intf_in_surf_boundary_pref`

Redefining Imported Features

About Redefining Imported Features

You can replace an imported feature with an import file, even if there is no one-to-one correspondence between the existing entities and the entities replacing them, using **Geometry** > **Reload File**. Select the imported feature in the Model Tree, right-click, and select **Edit Definition** on the shortcut menu.

You are prompted to pair all entities of the import feature referenced by other part and assembly features, both internal and external to the part. The edges, surfaces, and quilts are highlighted. You can use the external references for geometrical properties (for example, surface copies or composite datum curves) or spatial locators (for example, mating, aligning, or inserting), or both, geometrical properties and spatial locators.

You can set the correspondence between the referenced entities for all types of imported geometry using the **IMPORT PAIRS**, **SET PAIRS**, and **REROUTE FEAT** menus.

When replacing imported features,

- Use **Srf Analysis** to perform surface analysis on the original feature, especially when importing a CDRS IN feature.
- To redefine an imported feature from a file referenced externally in an assembly, set reference pairs manually with **Set Pairs**. You can set up pairs between references in the existing and the new imported feature. Some pairs are located automatically.
- Use **Auto Prompt** and **All Children**. These options step through each of the referenced entities to establish pairs for the new feature.
- Use **Set Pairs** only if the regeneration still fails after using **Auto Prompt** to find all missing references, and set appropriate pairs.

- Use the **REROUTE FEAT** menu to check the validity of the referenced pair only for the failed feature. Use **All Children** to check the validity of the reference pairs for all features.
- Do not pair the old references during initial pairing because each of the features referencing the old references can fail. Instead, pair the old references with the new references appropriate for that feature.
- You cannot roll back to features that existed before the imported feature using **Roll Back**. When you roll back a part to the imported feature, you can flip the direction of the feature to match the reference pairs for the direction of curves or the surface normal.
- Use **Current Feat** only when a reference in the old imported feature splits into many in the new imported feature, and you want to reference different references for different features in the new part. Each feature must map into a single correspondence.
- When redefinition is complete and regeneration successful, the old imported feature is regenerated and replaced with the new one. During the regeneration, you can click **Quit** on the **SET PAIRS** menu to delete the new feature and restore the original feature.

To Add a New Surface to an Existing Feature

1. Right-click to select the feature you want to redefine from the Model Tree. A shortcut menu appears.
2. Click **Edit Definition**.
3. Click **Geometry** > **Reload File**. The **File Open** dialog box opens.
4. Click the file type you want to import in the **Type** box. Files of the selected type located in the working directory are listed.
5. Select the file you want or browse to find the file.
6. Click **Open**. The **Layer Import Options** dialog box opens.
7. Close the Information window if it opens.
8. Make changes in the **Layer Import Options** dialog box as is necessary.
9. Click **OK**.
10. The **Choose Solid Options and Placement** dialog box opens.
11. Complete the dialog box and click **OK**.
    - The file with the data you want to reimport opens.
    - A Pro/ENGINEER window shows the original file.
    - The selected file appears in the main Pro/ENGINEER window.
    - The **IMPT FILE** menu appears with the following:

Replace All—Replaces the entire feature with the new import file.

Replace—Replaces selected parts of the feature with a new import file.

Add—Adds a new import file to the existing feature.

12. Click **Add** on the **IMPT FILE** menu.
13. Click **Done** on the **IMPT FILE** menu.
    - The existing feature is displayed in the main Pro/ENGINEER window with the new import file added to this feature.
    - The Pro/ENGINEER window that displayed the original file closes.

To Replace an Imported Feature

1. Add a new surface to an existing feature.
2. Click **Replace All** on the **IMPT FILE** menu.
3. Set up the new reference pairs by selecting commands on the **IMPORT PAIRS** menu which are as follows:
    - **Srf Analysis**—Displays the **CHOOSE REPL** menu.
    - **Ref Info**—Displays the **COMP REFS** menu with the following:

      Existing—Shows common references in the original and new features.

      New—Shows references in the new feature that are not present in the original feature.

      Missing—Shows references in the original feature that are not present in the new feature.

      The **SRF REFS** menu appears when you click **Existing**, **New**, or **Missing**. You can then show the pairs of references by clicking **Next** and **Previous**.
    - **Roll Back**—Rolls back the parts to any feature before setting pairs.
    - **Set Pairs**—Displays the **SET PAIRS** menu with the following options:

      Old Ref—Selects an old reference from one of the pairs that is located. To accept the corresponding new reference, click **Default Ref**.

      Default Ref—Accepts the default references that is selected.

      New Ref—Selects a new reference.

      Skip—Skips pairing an old reference.
    - **Auto Prompt**—Finds the references in the existing imported feature referenced by other features (these must be paired). You are prompted to select corresponding references in the new imported feature.
4. Click **Done**. The part is regenerated.

5. If the regeneration fails because of missing references, use the **IMPORT PAIRS** and the **REROUTE FEAT** menus to specify additional pairs.

   When redefinition is complete and regeneration successful, the old imported feature is regenerated and replaced with the new one.

   During the regeneration, click **Quit** on the **SET PAIRS** menu to delete the new feature and restore the original feature.

To Replace Part of an Existing Feature

1. Add a new surface to an existing feature.
2. Click **Replace** on the **IMPT FILE** menu. The **CHOOSE REPL** menu appears.
3. Select one of the commands on the **CHOOSE REPL** menu. The **REPLACE ENTS** menu appears. All types of entities are checked for replacement.
4. Select one of the following types of entities for replacement on the **REPLACE ENTS** menu:
   - **Point**
   - **Datum Plane**
   - **Curve**
   - **Surface**
5. Click **Done**. The system prompts you through the import process. After redefinition is complete and regeneration successful, the selected part of the feature is deleted and replaced with the new one.

Working with Imported Faceted Geometry

About Working with Imported Faceted Geometry

The tessellated files that are imported from STL, VRML, STEP, and MEDUSA contain faceted geometry.

You can use the imported faceted data in Pro/ENGINEER to:

- Calculate mass properties and measurements such as distance, length, angle, and area.
- Create datum features by referencing the faces, edges, and vertices of the faceted data.

You can select and reference imported faceted geometric entities just as you select and reference regular or exact geometry to create or redefine datum features such as points, planes, and axes.

The geometry is static and is an imported feature. Faceted geometric entities do not have permanent entity IDs as exact geometric entities. Referencing faceted data does not create parent-child relationships in the way that referencing exact geometry

does. The datum features remain at the same absolute location or orientation in the part they are created even if the faceted data is modified or moved.

There is no association between the faceted data and the datum features created by referencing the faceted data.

Note: The datum features are frozen in the model space of the parent model. Therefore, for an assembly with faceted component models, to update the datums with the faceted component location, create the datums in the context of the faceted component model. That is, activate the component and create the datum. The datums do not update to changes in the faceted geometry, but they update to changes in the component model location.

About Using Imported Faceted Geometry for Datum Creation

You can reference faces, edges, and vertices of the faceted data for the creation of datum features such as planes and axis. You can use facet geometry or a combination of facet and exact geometry as reference for the creation of datum features.

Create datum features referencing facet geometry as follows:

- **Datum plane**—Facet vertices, facet edges, and facet faces
- **Standard datum point**—Only facet vertices
- **Field datum point**—Only facet faces
- **Datum axis**—Facet faces, facet edges, and vertices

You can use normal, parallel, through, offset, and angular constraints for faceted references. If angle or offset constraints are used, you can only set the initial offset value or specify the initial angle. You cannot modify or change the angle or the offset value after the datum feature is created because the reference data is internally frozen at the initial creation location and the angle or offset parameters are not actually created.

There is no association between the faceted data and the datum features created by referencing the faceted data. The datum features are independent of the faceted geometric references and do not fail even if you delete the faceted data from the part, suppress, or redefine it.

References to faceted geometry are not stored or maintained in the feature. Any reference made to the faceted geometry creates an internally frozen reference that does not change. You can redefine, but not reroute, faceted references. During redefinition, all faceted references appear as frozen references that can be removed and replaced as required. For a combination of exact and faceted geometric references, you can edit the exact-geometry references and not the facet-geometry references. You can delete facet references and select new facet references when editing the definition of datum features.

About Creating Datum Axis Referencing Imported Faceted Geometry

You can create datum axis referencing faces, edges, and vertices as follows:

- Through a facet edge.
- Normal to a facet face. The datum axis is created at the point of selection, similar to a field point.
- Through two facet vertices.
- Through two facet faces.
- Through any combination of facet and exact geometric reference for the reference combinations listed above, for instance, through a facet vertex and a native vertex.

Creating Datum Points

You can create standard datum points at any facet vertex. Field datum points can be created on any facet face and are positioned at the pick locations.

About Creating Datum Planes Referencing Imported Faceted Geometry

You can create datum planes referencing imported faceted geometry or a combination of faceted and exact geometry as follows:

- Through a facet face.
- Offset from a facet face.
- Parallel to a facet face and through a facet vertex.
- Normal to a facet face and through two facet vertices.
- Normal to a facet face and through a facet edge.
- Through two parallel facet edges.
- Through a facet edge and a facet vertex.
- Through two facet vertices and angled to a facet face.
- Through three facet vertices.
- Through any combination of facet and exact geometric references for the combinations already listed, for instance, through two facet vertices and one real datum point.

About the Analysis of Imported Faceted Data

You can reference faces, edges, and vertices of the faceted data for the creation of datum features. You can also calculate the mass properties of faceted data and use faceted data for measurements.

You can use:

- **Analysis** > **Model Analysis** to analyze the mass properties of faceted data using the **Model Analysis** dialog box.
- **Analysis** > **Measure** to reference faces, edges, and vertices of the faceted data, create analysis features, and for measurements.

You can use the following measurement options on the **Measure** dialog box that opens when you click **Analysis** > **Measure**:

- **Curve Length**—Measures the length of the selected faceted edge. Use **Facet Edge**.
- **Distance**—Measures the distance with respect to the first reference entity selected. Select facet faces, facet edges, and facet vertices for distance measurement. Use **Facet Face**, **Facet Edge**, and **Facet Vertex**.
- **Angle**—Measures the angle between two selected entities. Select facet faces and facet edges for angle measurements. Use **Facet Face** and **Facet Edge**.
- **Surface Area**—Measures the entire surface area of an individual faceted feature. Select individual faceted B-reps. You cannot select single or groups of facet faces. You can only select the entire faceted B-rep of the import feature. Use **Facet**.

You must select valid faceted references in any combination for the calculation of distance and angle measurements. You can also mix faceted references with exact geometric references. For example, you can measure the distance between a facet face and a datum plane or a solid surface.

Note: For distance measurements, the facet faces and facet edges are treated as infinite planes and axes. A distance measurement between any nonparallel facet faces or nonparallel and coplanar facet edges returns a value of zero. This is different from the behavior when selecting finite surfaces and edges of exact geometry because these entities are treated as finite for the measurements (that is, unless they physically touch, the distance between them is not zero).

Exporting and Importing Layer Information

About Exporting Layer Information

You can export and import layers. To export layers, you can:

- Specify the layer IDs using **Specify Id** in the **SETUP LAYER** menu if the kind of export does not support named layers.
- Specify the layer name.

Some kinds of export support a single layer per entity. If an entity resides on more than one layer, only the highest layer ID (assigned the largest number) is exported as the layer of the entity.

The layer name for an entity read from the Neutral format is `INTF <nnn>`, where `<nnn>` is the layer ID.

You can export layer information for detail entities in the Drawing mode. However, the system does not output entities on blanked layers to the Neutral file.

If objects such as quilts, datum curves, and so on, have a blanked layer status, these objects are exported with their information blanked without altering the geometry of the model.

If assembly components are assigned individual layers in Pro/ENGINEER when you export assemblies to IGES in the flat mode:

- Interface IDs of layers are output as IGES levels. In an IGES file, assembly components are associated with levels instead of layers.
- If an assembly component is placed on a layer and blanked, all geometric entities of the blanked component are marked as invisible in an IGES file.
- If an assembly has a subassembly, to output layer information of the subassembly when the top-level assembly is output to IGES, you can assign a layer to the subassembly. At the subassembly level, place all the components of the subassembly on this layer, and assign an interface ID to the layer.

About Layer Export Status

You can:

- Select multiple layers and change their layer export status to the layer export status of a single selected layer.
- Export layers as numeric IDs or names.
- Provide an automated mechanism for placing assembly components on layers.
- Express an assembly structure by placing components on individual layers.

Not all of the Neutral file formats support layers with both IDs or names. The following table shows the layering functions that some of the file formats support. The **Choose Layers** dialog box lists these functions as follows:

	IGES	**VDA**	**STEP AP214_CD**	**CATIA**
Layer IDs	supported	not supported	not supported	supported
Layer Names	supported	supported (entity type set)	supported	not supported
Entities on Multi-Layers	supported	not supported	supported	not supported
Blanked Layers	supported	not supported	supported	not supported (emulated)

Assembly Components	supported	supported	supported	supported

For a part, the IGES and VDA formats support the export of sublayers. When exporting to STEP, entities on sublayers are placed on the top-level layers.

When exporting an assembly, if you change the layer export status of any one layer in the **Choose Layers** dialog box, all layers of the part reflect the same layer status.

About Controlling the Import and Export of Layers

You can control the import and export of layers and their entities during data transfer between Pro/ENGINEER and other CAD systems such as IGES, STEP, SET, VDA, CATIA, and Neutral.

You can set the import or export status of a given layer to one of the following:

- `Blank`—Marks as not visible the layers that are blanked. The entities residing on the blanked layers do not appear in the imported or exported model or assembly components.

  Note: When exporting layers, if you set the value of the `intf_out_blanked_entities` configuration option to `no`, blanked layers are not included for export.

- `Show`—Displays the layers with their entities. The layers with the entities are visible in the exported or imported model and assembly components.

- `Ignore`—Does not import or export the ignored layers. The ignored layers are not present in the imported or exported model or assembly components. However, the geometry residing on these layers is imported and exported. The geometry is not assigned layers or it is assigned the default layer in the imported or exported model.

- `Skip`—Does not import or export the skipped layers and their associated geometry.

To set a large number of layers to the same import status with a single setting, select all the layers you want to change using the standard CONTROL and SHIFT key multiple selection mechanism. After selecting the required layers, change the layer display status of one of the selected layers (the selected layers are highlighted) to the required status. This automatically updates the status of all the selected layers to the same status setting without you having to individually change the status of each layer.

Working with Data Exchange Formats

About Importing and Appending Files

You can create a Pro/ENGINEER model or drawing by importing a file using **File** > **Open**. To append a Pro/ENGINEER model or drawing with an imported file, use **Insert** > **Shared Data** > **From File**.

You can directly assemble imported files as new components of an existing Pro/ENGINEER assembly using **Insert** > **Component** > **Assemble**.

About Exporting Files from Pro/ENGINEER

You can export a Pro/ENGINEER model or drawing using **File** > **Save a Copy**.

About Drawing Sheet Size

The imported drawings are placed on the standard-size drawing sheet format in Pro/ENGINEER when the drawing sheet size of the imported drawings match the standard Pro/ENGINEER drawing sheet size.

For drawings in the IGES, STEP, DXF, DWG, MEDUSA, STHENO/PRO, CADAM, and CGM file formats, when the drawing sheet sizes vary from the standard Pro/ENGINEER drawing sheet size, drawing sheets of different sizes are automatically created in Pro/ENGINEER. This depends on the sheet size information contained in the imported file formats.

If the imported file does not contain the drawing sheet size information, the imported drawing is placed on a standard Pro/ENGINEER drawing sheet whose size is closest to the outline of the imported file.

When you import a DXF drawing file consisting of multiple drawing sheets of variable size, a single drawing is created in Pro/ENGINEER with multiple sheets corresponding to the number of drawing sheets contained in the DXF file. The variable sizes of the drawing sheets are maintained in Pro/ENGINEER or for each sheet of the drawing, a standard-size drawing sheet that is nearest in size to the original drawing sheet is created in Pro/ENGINEER.

About Custom Start Part and Assembly Template Files

You can import parts and assemblies into custom start part and assembly template files so that the imported models contain the predefined template information, such as datums, layers, saved views, and parameters as well as the imported geometry. You must set the value of the `intf_in_use_template_models` configuration option to `yes` to use start part and assembly template files when importing parts and assemblies through **File** > **Open**.

By default, Pro/ENGINEER uses the standard predefined start part and template files for import. However, if the `template_solidpart` and `template_designasm` configuration options are set to specify the path and name of the customized start part and assembly template files, these files are used when importing the part and assembly models.

If custom start part or start assembly template files are designated in the `config.pro` file, Pro/ENGINEER confirms the existence of these custom template files before performing the import. If Pro/ENGINEER cannot locate them, you are prompted whether to continue with the import. If you choose to continue, Pro/ENGINEER performs the import without using any template files.

Note: It is recommended that the custom start part template files contain only datum geometry.

ACIS

To Import a Part or Assembly from ACIS

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **ACIS File (*.sat)** in the **Type** box. The ACIS files in the working directory are listed.
3. Click the name of the file you want to import or browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Click **Part** or **Assembly**.
6. Click **OK** in the **Import New Model** dialog box.

To Export a Part or Assembly to ACIS

1. Open a part or assembly.
2. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
3. Select **ACIS File (*.sat)** in the **Type** box.
4. Accept the default name in the **New Name** box or type a new model or assembly name for the export.
5. Click **OK**. The **Export ACIS** dialog box opens.
6. Select the entities to export under **Export**.
    - **Solids**—Generates solids.
    - **Shells**—Generates shells.
7. Click **Customize Layers** in the **Export ACIS** dialog box. The **Choose Layers** dialog box opens.
8. Set the layer export status of a single layer or multiple layers. Click **OK** in the **Choose Layers** dialog box.
9. Use the default coordinate system or select or create a coordinate system for the part or assembly.
10. Click **OK**. If you are exporting an assembly, the system creates a file with the name `<name>_asm.sat`.

Adobe Illustrator Curves

About Importing Adobe Illustrator Curves

You can import Adobe Illustrator curves directly into Pro/ENGINEER as parts, assemblies, or sketches.

The following entities are ignored during the import of the Adobe Illustrator Curves:

- Color
- Fill
- Outline
- Grouping information

Note: You cannot import the Adobe Illustrator curves (file type `*.ai)` in the 2D modes, such as drawing, layout, diagram, and report.

To Import a Part, Assembly, or Sketch with Adobe Illustrator Curves

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **Adobe Illustrator (*.ai)** in the **Type** box. The Adobe Illustrator files in the working directory are listed.
3. Select the file you want to import or browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Select **Part**, **Assembly**, or **Sketch**.
6. Click **OK** in the **Import New Model** dialog box.

To Append a Part with Adobe Illustrator Curves to an Existing Model

1. Click **Insert** > **Shared Data** > **From File** to append a file with Adobe Illustrator curves as a part to an existing Pro/ENGINEER model.

   Click **Insert** > **Component** > **Assemble** to assemble a file into an existing assembly.

   The **File Open** dialog box opens.
2. Select **Adobe Illustrator (*.ai)** in the **Type** box. The Adobe Illustrator files in the working directory are listed.
3. Select the file you want to append or browse to find the file. The **Choose Solid Options and Placement** dialog box opens.

   The **Component Placement** dialog box opens if you are assembling a part component into an existing assembly.
4. Accept the default coordinate system location or select a coordinate system to position the geometry in the **Choose Solid Options and Placement** dialog box or the **Component Placement** dialog box.
5. Click **OK** in the **Choose Solid Options and Placement** dialog box or the **Component Placement** dialog box.

CADAM and CPTR Neutral Files

Importing CADAM Files

About Importing CADAM Files

You can use Pro/CDT to import Professional CADAM drawings containing draft entities and text into Pro/ENGINEER in addition to CADAM mainframe files in the CPTR-Neutral file format (`*.bin`). Pro/CDT, an optional Pro/ENGINEER module, provides the interface between Professional CADAM and Pro/ENGINEER.

You can directly transfer CADAM drawings when both CADAM and Pro/CDT are running on the same IBM RS/6000 machine. You can also import CADAM drawings by processing a CADAM file on one machine and retrieving it on a Pro/ENGINEER workstation. Install the Pro/ENGINEER `pro_from_cdm` utility on the same IBM RS/6000 workstation on which CADAM is running to process the CADAM file.

Using **File** > **Open**, you can directly import the Professional CADAM database. Pro/ENGINEER uses the default drawing setup information to determine how to handle dimensions and text. You can create a drawing setup file using the CADAM data setup information. For example, if CADAM data uses ISO parallel-type dimensioning, you can set the `text_orientation` drawing setup file option to `parallel`.

You can use Pro/DETAIL or Pro/LEGACY to modify the drawing imported from CADAM.

To Directly Import a CADAM Drawing

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Click **CADAM Database** in the **Look In** box.

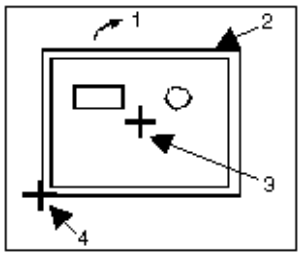
   Note: No list of files appears for the CADAM Database. You must know the full name of the file that you want to import.
3. Select **Cadam Direct (*.ext)** in the **Type** box.
4. Type the full path for the CADAM drawing that you want to import in the **Name** box.

   Note: The full path must include all spaces. For example,

   ```
   /CADAM/DRAWINGS/cad/cadam drawings/D cadam std inch
   ```
5. Click **Open** to append the CADAM file.
   - If the drawing sheet size varies from the standard Pro/ENGINEER drawing sheet size, a drawing sheet of appropriate size to fit the drawing, depending on the sheet size information contained in the file, is automatically created in Pro/ENGINEER.
   - If the drawing sheet size matches the standard Pro/ENGINEER drawing sheet size, the drawing is placed on the standard size drawing sheet.

Example: CADAM Drawings with Different Points of Origin

The following occurs when you type `Y` to move the left corner of a drawing to the screen origin:



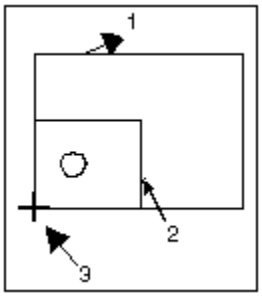
1 Screen

2 CADAM drawing

3 CADAM drawing origin

4 Pro/ENGINEER drawing origin

The following occurs when you type `N to` place the origin of a CADAM drawing, located in the center, at the origin of the Pro/ENGINEER drawing:



1 Screen

2 CADAM drawing

3 CADAM drawing origin placed at the Pro/ENGINEER origin

To Process a CADAM .cdm File

Create a file using the `pro_from_cdm` utility on an IBM RS/6000 on which the utility is installed and Professional CADAM is running. Type:

```
pro_from_cdm <group, subgroup, drawing> <outputfile>
```

where:

- `<group, subgroup, drawing>` is the CADAM name.
- `<outputfile>` is the file created with the name `<outputfile>.cdm.#`.

Note: Do not include spaces in the drawing name.

To Import the Processed .cdm File

1. Copy the processed `.cdm` file to a Pro/ENGINEER workstation.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **CADAM (*.cdm)** in the **Type** box.
4. Select the CADAM file you want to import from the list of available files, or type the name in the **Name** box.
5. Click **Open**. The **Import New Model** dialog box opens.
6. Complete the dialog box and click **OK**.

Importing Entities and Layers

The following table lists the entities and the layers on which they are placed when you import CADAM files to Pro/ENGINEER. Note that `<cadamoverlayname>` is the name of the overlay specified in CADAM.

Entity	Layer on Which Entity Is Placed
cadam_xyz_point	points_cdt_layer
cadam_2d_point	points_cdt_layer
cadam_nc	nc_style_cdt_layer
cadam_overlay	<cadamoverlayname>_olay

Tip: Transferring CADAM Dittos

A CADAM ditto is a separate part that you can instance at any point in a drawing. The instance can have any scale, angle of revolution, and hatches. A ditto instance exists as a set of independent entities in an imported drawing. These entities do not recognize the source of the ditto and are similar to other entities that are not part of any other ditto in the CADAM drawing.

You can use the `cdt_transfer_details` configuration option to determine the dittos in a CADAM drawing. When set to `yes`, the details associated with a CADAM drawing are converted to additional sheets on the drawing.

Importing CPTR-Neutral Format Files

About Importing CPTR-Neutral Format Files

You can import CADAM mainframe files using the CPTR-Neutral file format (`*.bin`) besides importing CADAM drawing files. A converter supplied by Integrated Industrial Information, Inc., CPTR, converts CADAM mainframe data to the CPTR-Neutral format by way of ACCESS:IUE. Using Pro/CDT, the optional Pro/ENGINEER module that provides the interface between Professional CADAM and Pro/ENGINEER, .you can then import a CPTR-Neutral file into Pro/ENGINEER.

You can import a CPTR-Neutral file in either of the following ways:

- Directly on a computer that has Pro/CDT and pro_from_cptr installed.
- Process the CPTR-Neutral file on one machine and retrieve it on another machine by:
    - Running the Pro/ENGINEER pro_from_cptr utility to convert the the CPTR-Neutral (*.bin) file into a .cdm file .that can be read by Pro/CDT.
    - Importing the .cdm file into Pro/ENGINEER.

Note: Import of the CPTR-Neutral file format is not supported on Linux.

CADAM Font Table characters (ITYPE 1060) are supported as instances of user-defined Pro/ENGINEER symbols. A CPTR-Neutral file can contain the CADAM element, Plot Axis System (ITYPE 960), defining the format and positioning of a CADAM drawing in Pro/ENGINEER.

The use_cadam_plot_data configuration option specifies whether to define the information in the Plot Axis System element when you import a CADAM drawing into Pro/ENGINEER. If the option is set to no (the default), the system ignores it. If it is set to yes, and the imported drawing contains the Plot Axis System element, you are asked if you want to define the format based on the Plot Axis System.

- Type Y, to select the format based on the height, width, and origin defined by the Plot Axis System element.
- Type N to select the format of the regular CADAM files.

You can use the cadam_line_weights configuration option to define the line width of entities and plot drawings with correct line weights that are consistent with a standard.

To Directly Import a CPTR-Neutral File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **CADAM CPTR Neutral (*.bin)** in the **Type** box.
3. Select the CADAM file you want to import from the list of available files, or type the name in the **Name** box.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Complete the dialog box and click **OK**.

   If you set the use_cadam_plot_data configuration option to yes, and the imported drawing contains the Plot Axis Syst Step 7em element, you are prompted to define the format based on the Plot Axis System.
6. Type Y, to complete the import process or type N if the imported drawing does not use the Plot Axis System.

   If the imported drawing is larger than the selected standard size format, you are prompted to scale down the drawing to fit the format.
7. Type Y to scale the drawing.

The system asks if you want to move the left corner of the drawing to the screen origin.

8. Type `Y` to select the closest standard drawing size to fit the drawing and display the entire drawing.

   Type `N` to place the origin of the CADAM drawing, located in the center, at the origin of the Pro/ENGINEER drawing, located in the lower-left corner of the screen.

   The format is selected automatically, but the drawing is only partially displayed. To display the entire drawing or its portion on the screen, use **Zoom Out**, then translate entities by specifying a translation vector.

To Process a CPTR-Neutral File

Convert a CPTR-Neutral file to a `.cdm` file by running the Pro/ENGINEER utility, `pro_from_cptr`. On a machine with `pro_from_cptr` installed, type:

```
[pro_from_cptr <input_file> <output_file>]
```

where:

- `<input_file>` is the name of the CPTR-Neutral file.
- `<output_file>` (optional) is the name of the output file. If the output file name is not specified, the system adds the extension `.cdm` to the file name.

To Import the Processed CADAM File

1. Copy the processed CADAM file to a Pro/ENGINEER workstation.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **CADAM (*.cdm)** in the **Type** box.
4. Select the CADAM file you want to import from the list of files, or type the name in the **Name** box.
5. Click **Open**. The **Import New Model** dialog box opens.
6. Complete the dialog box and click **OK**.

Importing Cross-Hatch Patterns

You can import cross-hatch patterns into Pro/ENGINEER using the CADAM Areafill entity (ITYPE=1600). Hatches are not supported during the import unless CADAM files contain hatch information. The following table shows the correspondence between CADAM and standard Pro/ENGINEER cross-hatch patterns.

Display CADAM Pattern Name	Pro/ENGINEER Pattern Name
ALUM1	`aluminum.xch`

COPPER	copper.xch
TILE	electric.xch
DASHES	glass.xch
HATCH	iron.xch
RUBBER	plastic.xch
STEEL	steel.xch
TITANIUM	titanium.xch
GRID	zinc.xch

If a CADAM cross-hatch pattern does not have a corresponding standard Pro/ENGINEER pattern, you can do one of the following:

- In Pro/ENGINEER, create a file with a user-defined cross-hatch pattern and store it in the working directory or the `<loadpoint>/text/crosshatch area`. The name of the cross-hatch file must be `cdm_<NAME>.xch`, where `<NAME>` is the name of the CADAM cross-hatch pattern to be imported. For example, if the name of a CADAM Areafill pattern is `ALUM2`, the corresponding name of a user-defined pattern must be `cdm_alum2.xch`. On import, the CADAM pattern is replaced with the user-defined pattern of the same name.
- Let the system create a pattern. If a corresponding user-defined pattern is not found, the system creates a hatch using a simple hatch pattern, for example, slanted lines.

CADDS 5

Overview of Pro/INTERFACE for CADDS

About Pro/INTERFACE for CADDS 5

With Pro/INTERFACE for CADDS, you can import CADDS parts and assembly structures into Pro/ENGINEER and export Pro/ENGINEER parts and assemblies to native CADDS format.

You can transfer the following types of 3D geometry between Pro/ENGINEER and CADDS 5: solid models, assembly structures, surfaces, wireframe curves, and points. The resulting part geometry is merged into a single import feature and there is no transfer of feature history or parametric relationships.

You cannot transfer two-dimensional (2D) CADDS or MEDUSA drawing data with this interface. The following CADDS 2D drawing entities are not supported:

- Nodal Subfigures
- Nodal lines

- Text
- Drawings
- Views
- Dimensions
- Labels

Note: You can use the CADDS 5 interface options only on the Silicon Graphics, Sun, or Hewlett-Packard platforms.

Associative Topology Bus and CADDS 5

CADDS 5 is an ATB-enabled data exchange format. ATB allows the associative update of imported data when changes are made to the native CADDS 5 data.

For information, see the Help on Associative Topology Bus.

About Defining Components by Path Location

In CADDS 5, you can define components of assemblies by explicitly building their path locations into the component. For each CADDS 5 assembly component node, you can designate a model by one of the following methods:

- Model name only, such as `part1`, where the CADDS 5 component must exist in a valid search path.
- Path location and model name, such as `/proj1/revision3/part1`, where the entire directory structure must exist in a valid search path.

For example, when the component model designation for a Concurrent Assembly Mock-up (CAMU) assembly node is `/proj1/revision3/part1` and your Pro/ENGINEER session is set up as follows:

- The `CVPATH` environment variable is set to `/home/user1/old-parts`
- The `extend_cvpath` configuration option is set to `/home/user1/new-parts`

The following are valid locations for the imported `part1` CADDS 5 model:

- `/proj1/revision3/part1`
- `/home/user1/old-parts/proj1/revision3/part1`
- `/home/user1/new-parts/proj1/revision3/part1`

The embedded path of the component model definition must start relative to a defined search path or be a complete system path such as `/proj1/revision3/part1`. You cannot find `part1` in `/home/user1/old-parts/part1` or in `/home/user1/new-parts/part1`. Pro/ENGINEER searches for `/proj1/revision3/part1` in the defined search path, not `part1` alone.

If the component designation of the CAMU node is the CADDS 5 model name only (`part1`), then both `/home/user1/old-parts/part1` and `/home/user1/new-parts/part1` are valid locations.

About Setting the CVPATH Environment Variable

The CVPATH environment variable is used by the CADDS 5 translator to define the following:

- Search paths for locating components of CADDS 5 Concurrent Assembly Mock-Up (CAMU) assemblies selected for import.
- Search paths for locating CADDS 5 part or assembly reference models when using Associative Topology Bus (ATB) **Check Status** and **Update** commands.
- Location on the hard disk where CADDS 5 models exported from Pro/ENGINEER is written.

You must set the CVPATH environment variable before you start your Pro/ENGINEER session to define the default paths for the CADDS models. Use the extend_cvpath configuration option to add additional paths to the CVPATH definition during a Pro/ENGINEER session using **Tools** > **Options**. These additional paths are added to the end of the existing CVPATH environment variable setting.

Note: The extend_cvpath configuration option setting does not actually change the CVPATH environment variable setting. The search paths are added during runtime of the Pro/ENGINEER session.

You can set as many directory paths as you want. Use the following conventions when setting the CVPATH:

- Paths may be relative to the Pro/ENGINEER startup directory, or absolute paths of the hard drive.
- The paths are contained within single quotes.
- Each path is separated by a colon (:).
- The paths are defined in the required search order. When searching for CAMU assembly component models or ATB reference models, the search starts with the first path and traverses the paths sequentially until the model is located.
- You can assign the =C to the creation directory of the CADDS 5 export files in the CVPATH. The creation directory is where the CADDS models from Pro/ENGINEER are created on export.

  Note: The creation directory overrides the **Save a Copy** dialog box setting. The export location set in the **Save a Copy** directory is ignored for the CADDS 5 format.

An example of how to set the CVPATH environment variable follows:

```
setenv CVPATH '<path1>=C:<path2>:<path>:...<pathx>'
```

The =C after the <path1> in the example, indicates that this path is the creation directory.

Pro/ENGINEER searches for CADDS components in the current working directory or uses the CVPATH environment variable for locating components in other directories.

The additional paths, added to the end of the existing CVPATH environment variable setting, are not searched until after the paths set in the actual environment variable

are searched. You cannot set the creation directory with the extend_cvpath configuration option.

To retrieve a CADDS assembly in Pro/ENGINEER, you must make sure that the CVPATH is set to the appropriate directories. When importing a CADDS 5 assembly, if the existing part or subassembly components are stored in different directories, designate each directory in the path for a successful import. Otherwise, an assembly with empty Translated Image Model (TIM) parts is created.

Note: If the CVPATH environment variable is not explicitly set, Pro/ENGINEER uses the default setting of a single search path according to the current working directory in Pro/ENGINEER. This directory is also used as the CADDS 5 export creation directory.

About the ODB_DAEMON Process

When you import a CADDS 5 Concurrent Assembly Mock-Up (CAMU) assembly or export a Pro/ENGINEER assembly to the CADDS 5 format, an ODB_DAEMON process is started. This process is not started or required for the import and export of a CADDS 5 part.

This process is generally not killed after the import of a CAMU assembly or the export of a Pro/ENGINEER assembly to the CADDS 5 format. It is directly linked to the CADDS 5 converter version and the specific geometric libraries required for the import of a CAMU assembly or the export of a Pro/ENGINEER assembly to the CADDS 5 format. If an ODB_DAEMON process is already running, then the existing ODB_DAEMON process is used during the import or the export of the assembly instead of starting a new ODB_DAEMON process.

Because multiple converter versions for CADDS 5 exist, and you can switch between these converter versions using the intf_cadds_version configuration option, you must check if an ODB_DAEMON process exists and is running before you import or export the assembly. If the version of the CADDS 5 converter, to which the existing ODB_DAEMON process is linked, does not match the converter version which is currently set by the intf_cadds_version configuration option, the import or export of the assembly fails. To avoid this, kill all existing ODB_DAEMON processes and start a new ODB_DAEMON process before you import the CAMU assembly or export the Pro/ENGINNER assembly to the CADDS 5 format.

Note: The ODB_DAEMON process continues to run even when you exit the Pro/ENGINEER session and can affect the behavior of Pro/ENGINEER sessions in the future unless you kill the ODB_DAEMON process after the import or export of the assembly.

About the Name Service

The Name Service is a process that is used by Pro/ENGINEER and invoked when Pro/ENGINEER starts. It must be running as follows for the CADDS 5 converter to work successfully:

```
<Pro/E Loadpoint>/<Platform Type>/obj/nmsd -timeout 300
```

The Name Service is started at the beginning of the Pro/ENGINEER session. It handles communication between Pro/ENGINEER and other separate executables that Pro/ENGINEER interacts with, such as the CADDS 5 converter. If the Name Service is not used during a session of Pro/ENGINEER, it times out five minutes after the start of the session. If it times out, the Name Service must be manually restarted, or a fresh Pro/ENGINEER session must be started to successfully use the CADDS 5 interface.

Importing CADDS 5 Parts and Assemblies

About Importing CADDS 5 Models

You must set the intf_cadds_version configuration option to 12 or 13 to import CADDS 5 parts and assemblies.

Import of CADDS 5 parts and assemblies to Pro/ENGINEER is backward compatible. For example, if you set the pro_cadds5 converter to CADDS 5 version 12, you can only export Pro/ENGINEER parts and assemblies to CADDS 5 version 12. However, you can import parts and assemblies into Pro/ENGINEER from CADDS 5 version 12 and earlier CADDS 5 versions such as CADDS 5 version 11, 10, and so on.

You can import the following CADDS entities to Pro/ENGINEER:

Hyperbola	Parabola	Nspline	Plane	Rsurface	Srev
Shell	Uv-Curve	Point	Line	Arc	Ellipse
Tcyl	Nsurface	Brep	Edge	Face	Loop

When you import CADDS parts and assemblies into Pro/ENGINEER, the layers along with the associated colors are also imported. The colors assigned to each CADDS layer are transferred to the corresponding entities in the imported model. If there are changes to the layers, or the colors associated with the layers, you can use Associative Topology Bus (ATB) to update the imported model.

Pro/ENGINEER regards imported parts and assemblies, including all of their components, as associative with the referenced parts and assemblies and owned by the referenced system.

To Import a CADDS 5 Part or Assembly

1. Set the CVPATH environment variable, if required.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Click the **Type** arrow to see more options.
4. Select **CADDS5** in the **Type** box. The browser defaults to one of the paths defined in your CVPATH environment variable. This path appears in the **Look In** box.

   Note: If the CVPATH is not defined, the browser, by default, displays the contents of the current Pro/ENGINEER working directory.

5. Click the **Look In** arrow to display a list of the paths defined in the `CVPATH` environment variable.
6. Select the path from which you want to import a part or assembly.
7. Double-click the name of the CADDS part or assembly you want to import.
8. The **Import New Model** dialog box opens. **Part** or **Assembly** is automatically selected.
9. Type a name for the imported part or assembly or accept the default name in the **Name** box.
10. Click **OK**.

Exporting from Pro/ENGINEER to CADDS 5

About Exporting to CADDS 5

When you export a part or assembly from Pro/ENGINEER to CADDS 5, you are working in the Explicit mode of CADDS, an industry-standard CAD/CAM design environment. The Explicit mode allows you to access the various CADDS applications.

Edge and surface definitions are transferred from Pro/ENGINEER to CADDS 5 in their exact geometric form using the Non Uniform Rational B-Splines (NURBS) mathematical description.

Execute File

When you export an assembly from Pro/ENGINEER to CADDS 5, an execute file named `<export model name>-exec` is created. When you run the execute file with CADDS 5, it creates:

- Default drawings and views for the exported CADDS models.
- Temporary view files (tvfs) that you can use to view CADDS parts in an assembly.
- If Associative Topology Bus is enabled when you are exporting an assembly, and the assembly contains cabling geometry, the execute file converts this exported cable data to CADDS 5 cables.

To Export a Part or Assembly to CADDS 5

1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **CADDS5** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. Accept the default name or type a new model name in the **New Name** box.
4. Click **OK**. The Pro/ENGINEER part or assembly is exported to CADDS 5 in the Explicit format.

CATIA

Overview of Licenses and Translators

About CATIA Versions, Licenses, and Translators

Pro/ENGINEER supports CATIA versions 4 and 5. The licenses required for the exchange of data between these versions of CATIA and Pro/ENGINEER are as follows:

- CATIA II Interface license for CATIA V4
- The CATIA V5 interface license (`intf_for_catia_v5`)

Importing and Exporting CATIA V4 Files

The CATIA V4 translators that are available in Pro/ENGINEER are as follows. The one which is used depends on the availability of the CATIA II Interface license:

- Interface for CATIA II with ATB when the CATIA II Interface license is available
- Pro/INTERFACE for CATIA when the CATIA II Interface license is not available

The availability of the CATIA II Interface license also determines whether CATIA V4 models imported into Pro/ENGINEER are ATB-enabled or not.

Note: Associative Topology Bus (ATB) functionality does not support the Pro/INTERFACE for CATIA translator.

The Interface for CATIA II with ATB

If you have the CATIA II Interface license, the Interface for CATIA II with ATB translator is used by default for the direct import and export of CATIA V4 files. CATIA V4 parts and assemblies imported to Pro/ENGINEER with ATB enabled can be associatively updated to changes in the native CATIA models with the Interface for CATIA II with ATB. For more information, see the Help module, Associative Topology Bus.

Note: You must purchase the CATIA II Interface license to use the ATB capabilities on the CATIA V4 models imported to Pro/ENGINEER.

Pro/INTERFACE for CATIA

If you do not have the CATIA II Interface license, Pro/INTERFACE for CATIA is used for the direct, one-step import and export operations. Pro/INTERFACE for CATIA does not support ATB. You cannot associatively update geometry imported with the Pro/INTERFACE for CATIA translator to changes in the original, native CATIA models.

Importing CATIA V5 Files

You must purchase the CATIA V5 interface license, `intf_for_catia_V5`, to import CATIA V5 part files (`*.CATPart`) to Pro/ENGINEER. You can only import CATIA V5 R6 and higher `*.CATPart` files to Pro/ENGINEER.

The CATIA V5 interface supports ATB. You can associatively update CATIA V5 geometry imported with the `topobus_enable` configuration option set to `yes`.

About Interface for CATIA II with ATB

The Interface for CATIA II with ATB is used by default to exchange geometry between Pro/ENGINEER and CATIA when the Interface for CATIA II with ATB license is available. The Interface for CATIA II with ATB does not require a CATIA installation and does not use the CATIA CATGEO API. It is supported on Sun, HP, SGI, and Windows.

The Interface for CATIA II with ATB supports the following:

- Import and export of CATIA V4 Model (`*.model`) files
- Import of CATIA V4 Model (`*.exp`) and CATIA V4 Session (`*.session`) files
- Model Space parameter configuration options
- Export to CATIA start parts
- Export of faceted data
- Import of CATIA layer filters

About Pro/INTERFACE for CATIA

Pro/INTERFACE for CATIA comes free with Pro/ENGINEER Foundation. This translator requires a CATIA installation to perform a complete transfer of data between Pro/ENGINEER and CATIA and uses the CATIA CATGEO API. It is supported on Sun, HP, SGI, and IBM (32-bit only) workstations.

Geometry is transferred between Pro/ENGINEER and CATIA by the following methods:

- One-step or the direct method
- Two-step or the indirect method

Regardless of whether you are using the one-step or the two-step method of data exchange or whether CATIA and Pro/ENGINEER are on the same workstation, the following utilities are used during file conversion:

- `pro_from_cat4`
- `progeo_to_cat4`

When CATIA and Pro/ENGINEER reside on different machines, you must copy the Pro/INTERFACE for CATIA utilities on a workstation which has CATIA installed and has access to CATIA licenses.

Pro/INTERFACE for CATIA does not support Associative Topology Bus (ATB).

About the Methods of Data Exchange Between Pro/ENGINEER and CATIA V4

With the Pro/INTERFACE for CATIA translator, data exchange between Pro/ENGINEER and CATIA V4 follows a one-step or a two-step method. CATIA and Pro/ENGINEER can reside on different workstations or on the same workstation.

Two-Step or the Indirect Method

You must have CATIA 4.2.0 on an IBM RS/6000, SGI, Sun, or Hewlett Packard workstation and Pro/ENGINEER on any other workstation. Pro/INTERFACE for CATIA must be available on the CATIA workstation.

Data exchange between Pro/ENGINEER and CATIA V4 involves the conversion of Pro/ENGINEER and CATIA V4 files to the intermediate `.ct` file format. You must manually start the `pro_from_cat4` and `progeo_to_cat4` utilities as a `catusr` to complete the file conversion. The `pro_from_cat4` and `progeo_to_cat4` utilities must be available on the workstation with CATIA installed and are used as described:

- `pro_from_cat4`—For import of CATIA V4 files to Pro/ENGINEER
- `progeo_to_cat4`—For export of Pro/ENGINEER files to CATIA V4

The `pro_from_cat4` utility converts the CATIA V4 `.model` file to a `.ct` file when you import CATIA V4 files to Pro/ENGINEER. After the conversion, you must copy the `.ct` file to the Pro/ENGINEER workstation.

For export, create the `.ct` file on the Pro/ENGINEER workstation and then copy the `.ct` file to the CATIA workstation. Use the `progeo_to_cat4` utility to convert the `.ct` file to a `.model` file.

To use the utility `progeo_to_cat4`, CATIA solids must be available on the receiving side. You can run this utility in default mode or user input mode.

One-Step or the Direct Method

You must have Pro/ENGINEER and CATIA 4.2.0 running on the same IBM RS6000, SGI, Sun, or Hewlett Packard workstation.

The CATIA V4 `.model` files are automatically converted to `.ct` files for import and the `.ct` files are automatically converted to `.model` files during the export. The `pro_from_cat4` and `progeo_to_cat4` utilities are used during file conversion when you select **CATIA V4 ProGEO Model (*.model)** from the **Type** list during import and export. You must run Pro/ENGINEER as a `catusr`. The system extracts data directly from the CATIA database.

Using Interface for CATIA II with ATB

About Importing from CATIA Using Interface for CATIA II with ATB

You can import the following types of CATIA files:

- CATIA V4 Model (`*.model`) part or assembly files

- CATIA V4 Session (`*.session`) assembly files
- CATIA V4 Model (`*.exp`) export files

You can filter and selectively import the `.model` files from the CATIA V4 Model `.exp` file. The `.model` files within a `*.exp` file are imported as part components of a native assembly in Pro/ENGINEER.

Note: You cannot create a single independent part from a `.model` file of a `.exp` file unless the `.exp` export file contains a single `.model` file which is automatically imported as a part.

You can import a CATIA V4 model with dittos as an assembly or a part. Dittos are used to represent assembly structures. Therefore, the default import model type for models with dittos is assembly.

You can open CATIA V4 files as new part or assembly models or append them to existing models in Pro/ENGINEER.

CATIA supports Associative Topology Bus (ATB). CATIA models, except `.exp` files, are imported as Translated Image Models (TIMs). Only CATIA files imported as new models using **File** > **Open** or assembled as a new component of an existing Pro/ENGINEER assembly are ATB-enabled. CATIA import features that are inserted as features to existing Pro/ENGINEER part files are not ATB-enabled.

You can use the ATB capability to update the imported geometry associatively when there are changes in the CATIA reference files. For information on ATB, see the Help on Associative Topology Bus.

Entities Supported for Import with Interface for CATIA II with ATB

You can import the following CATIA entities to Pro/ENGINEER:

- Faces
- Surfaces
- 2D entities (lines, circles, curves, and so on)
- Model edges
- Composite curves
- Points
- 3-axis entities (imported as coordinate systems)
- Datum planes
- Layers
- Layer filters

Importing CATIA V4 Models with Dittos in Assembly Mode

When you import a CATIA V4 model with dittos in the Assembly mode, model data is transferred to Pro/ENGINEER according to the following rules:

- You are prompted for the name of the assembly in Pro/ENGINEER. The CATIA V4 model is then imported into this assembly.
- For the Master Workspace, a subassembly with the name of the CATIA V4 model is created. In addition, a part file containing geometry that exists directly in the Master Workspace is created and named as the Master Workspace.
- If the Master Workspace or any other Detail Workspace does not reference the geometry, this Detail Workspace is ignored.
- For each Detail Workspace used in the Master Workspace, that is not used in any other Detail Workspace, a part file with the name of this Detail Workspace is created.
- For each Detail Workspace used in the Master Workspace that contains dittos, a subassembly with the name of the Detail Workspace is created. If the Detail Workspace contains additional geometry that is not from a ditto, this geometry is stored in a part file as a component of the subassembly with the name of the Detail Workspace.
- Each part component of the assembly is imported with the layer filters of the CATIA V4 master model.

Importing CATIA V4 Models with Dittos in Part Mode

You can import CATIA V4 models with dittos in Part and Assembly modes.

When you import a CATIA V4 model in the Part mode, the resulting part file contains the merged geometry of all its workspaces.

To Import a CATIA V4 Model File Using Interface for CATIA II with ATB

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **CATIA V4 Model (.model, .exp)** in the **Type** box.
3. Select the CATIA V4 file you want to import from the list of available files or browse to find the CATIA file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Click **OK**.

To Append a CATIA V4 File to an Existing Model Using Interface for CATIA II with ATB

1. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
2. Select **CATIA V4 Model (*.model, *.exp)** in the **Type** box.
3. Select the CATIA V4 file you want to import from the list of available files or browse to find the file.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.

5. If the target model contains coordinate systems, select one to locate the geometry of the import feature or select the default location.
6. If you are appending file to a Pro/ENGINEER part file with solid geometry, select **Protrusion**, **Cut**, or **Surfaces** to add to the solid geometry, subtract from the solid geometry, or represent the imported geometry as surfaces, leaving the solid geometry unchanged, respectively.
7. Click **OK**.

To Selectively Import Model Data of a CATIA V4 Export .exp File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **CATIA V4 Model (.model, .exp)** in the **Type** box.
3. Select the CATIA `*.exp` file you want to import from the list of available files or browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens. **Assembly** is automatically selected by default.

   Note: You can import the `.model` files within a `*.exp` file only as part components of a native Pro/ENGINEER assembly.
5. Click **OK**. The **Layer Import Options** dialog box opens. It lists each `.model` file contained within the `.exp` file. By default, the layer import status of all the .model files is **Show**.
6. To filter models that you do not require, select the model from the layer tree and set its import status to **Skip**.
7. Click **OK**. The `.model` files with their import status set to **Show** are imported.

About Importing a CATIA V4 Model with Layers and Layer Filters Using Interface for CATIA II with ATB

You can import a CATIA V4 model along with its layers and layer filters to Pro/ENGINEER and use the display status of the imported CATIA layers and the layer filters to selectively import geometry from the CATIA layers.

The layer filters of the imported CATIA V4 model are converted to master layers in Pro/ENGINEER. All layers assigned to a layer filter in CATIA are assigned to the corresponding master layer in Pro/ENGINEER. The association of the master layers with their assigned layers in Pro/ENGINEER is the same as the corresponding association of the layer filters with their assigned layers in CATIA. The master layers and their layers in Pro/ENGINEER retain the original names of the layer filters and their layers in CATIA.

Note: If Associative Topology Bus is active, you cannot filter layers when you import CATIA V4 models.

About Using Layer Display Status for Selective Import of CATIA V4 Geometry

You can use the display status of the imported CATIA layers and the layer filters to display or hide the imported entities residing on these layers and selectively import geometry from the CATIA layers. Use the **Layer Import Options** dialog box to set the initial display status of the imported layers and filter geometry residing on the layers.

At any given time, you can only set the display status of the active layer filter. If you save the setting of the active layer filter in the CATIA V4 model file before import, by default, the display status of the active master layer in Pro/ENGINEER is **Isolate**. All layers assigned to this master layer are displayed in Pro/ENGINEER.

About Exporting to CATIA V4 Using Interface for CATIA II with ATB

You can export exact surface and solid geometry to a CATIA V4 Model `.model` file. You can also export facet data and datum entities, such as points and curves of the Pro/ENGINEER part or assembly.

You can use the **Export CATIA** dialog box to customize the export of a Pro/ENGINEER model to CATIA. You may choose to export exact solid geometry as wireframe edges, surfaces, shells, or solids, depending on type of representation you want to export. You can also include or exclude facet geometryand datum entities.

When exporting an assembly, you may also choose one of the following for the exported assembly structure:

- **Flat**—Exports the assembly to a single master workspace of a CATIA V4 Model `.model` file. Each assembly component is exported as a separate volume. All the volumes of the assembly merge into a single master workspace. This is the default behavior.
- **Dittos**—Preserves the structure of multiple-level assemblies in the CATIA V4 Model `.model` file. The Pro/ENGINEER assembly components are exported as CATIA dittos or volumes with a workspace for each volume into a single CATIA V4 Model `.model` file.

If the CATIA V4 model selected for export already exists, use the `catia_out_to_existing_model` configuration option to control conflict resolution. You can append data to the existing CATIA V4 model or overwrite the existing model.

To Export to CATIA V4 Using Interface for CATIA II with ATB

1. In an open part or assembly, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **CATIA V4 Model (*.model)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export CATIA** dialog box opens.

5. Complete the **Export CATIA** dialog box to specify the structure and contents of the output file.

   If you are exporting an assembly, select **Flat** or **Dittos** from the **File Structure** list to specify the structure of the assembly.

6. Click **Customize Layers** from the **Export CATIA** dialog box to export layers. The **Choose Layers** dialog box opens.

7. Click **Auto ID** to manually assign layer IDs to layers that you want to export, but do not currently have IDs.

   You can use the **Choose Layers** dialog box to skip geometry on certain layers.

8. Click **OK** in the **Choose Layers** dialog box.

9. Click **OK** in the **Export CATIA** dialog box.

About CATIA Start Parts

You can use custom CATIA V4 `.model` start parts when exporting Pro/ENGINEER parts or assemblies to the CATIA V4 `.model` file format. The custom start-part template files define nongeometric entities such as workspace, views, layers, fonts, colors, graphic standards, tolerance, and other parameters.

The attributes of the start parts are applied to the exported CATIA V4 `.model` files. The following parameters from the CATIA start-part template file are copied to the Pro/ENGINEER models:

- Graphics standards
- Colors
- Tolerances
- Thickness
- Model comments
- Initial view plane
- Project file information

To access and use the CATIA start-part template files the `intf_out_cat_start_model` configuration option must be set to the path of the CATIA start `.model` file. If the start parts are missing, the exported CATIA V4 `.model` files use the default model parameter values.

About Controlling Model Space Parameters When Exporting to CATIA

For the exported CATIA V4 `.model` file to open correctly in CATIA, you must ensure the accuracy of the geometric data in the exported file by setting accuracy values. Explicitly set these values in the exported `.model` file through the following configuration options:

- `intf3d_out_cat2_model_sz`

- intf3d_out_cat2_ident_crv
- intf3d_out_cat2_ident_pt
- intf3d_out_cat2_infinity
- intf3d_out_cat2_sag
- intf3d_out_cat2_step

You can set any value or the recommended value that is based on a standard Model Size. The recommended tolerance values are based on the correlation between a standard Model Size and the parameters.

If you assign values other than the recommended values and open the exported Pro/ENGINEER part in CATIA, an error message displays the current tolerance value against the recommended values.

The following table shows the configuration options with their values in CATIA for the corresponding parameters in Pro/ENGINEER and CATIA. The default values for the parameters are based on the standard Model Size of 10000.

Parameter in Pro/ENGINEER	Model Data in CATIA	Configuration Option	Recommended Values in CATIA
Model Size	Model Size	intf3d_out_cat2_model_sz	10000
Tolerance Curve	Identical Curve	intf3d_out_cat2_ident_crv	0.1
Tolerance Point	Intersection Projection	intf3d_out_cat2_ident_pt	0.001
Tolerance Line	Infinity	intf3d_out_cat2_infinity	100000
Tolerance Sag	Bending	intf3d_out_cat2_sag	0.03
Tolerance Distance	Step	intf3d_out_cat2_step	20

If you set the model space parameter configuration options and also use the CATIA start parts, the model space parameter values of the designated CATIA start parts override the values of the configuration options.

Using Pro/INTERFACE for CATIA

To Convert a CATIA .model File to the Neutral .ct Format

Perform the following steps if you do not have Pro/ENGINEER and CATIA on the same workstation:

1. Copy the pro_from_cat4 utility to the workstation on which CATIA is installed.
2. On the CATIA workstation, run pro_from_cat4 utility using one of two methods:

Type `pro_from_cat4` and interactively specify input values or accept the defaults.

or

Type all inputs in command-line format, as follows:

```
pro_from_cat4 [-n name] [-d ddname] [-m model_path] [-p project] [-o filename] [-v verbose] [-r revision_number]
```

Modifiers are similar to those of the `progeo_to_cat4` utility. The extension `.ct` is appended to the new file if you do not specify the `-o <filename>` modifier when executing the utility.

The default is to place the model in the `/home/catusr/db` directory and to run the script from the `/home/catusr` directory.

3. Copy the processed CATIA file to a workstation with Pro/ENGINEER installed.

To Import a Neutral .ct File

1. Copy the processed CATIA file to a workstation with Pro/ENGINEER installed.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **CATIA to/from File (*.ct)** in the **Type** box.
4. Select the CATIA file that you want to import from the list of available files or type the name in the **Name** box.
5. Click **Open**. The **Import New Model** dialog box opens.
6. Complete the **Import New Model** dialog box and click **OK**.

To Append a Neutral .ct File to a Pro/ENGINEER Part

1. Copy the processed CATIA `.ct` file to a workstation with Pro/ENGINEER installed.
2. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
3. Select **CATIA to/from File (*.ct)** in the **Type** box.
4. Select the CATIA file from the list of available files or browse to find the CATIA file.
5. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.
6. If the target model contains coordinate systems, select a coordinate system to locate the geometry of the imported feature or select the default location.
7. If you are appending the file to a Pro/ENGINEER part file with solid geometry, select **Protrusion**, **Cut**, or **Surfaces** to add to the solid geometry, subtract from the solid geometry, or represent the imported geometry as surfaces, leaving the solid geometry unchanged, respectively.
8. Click **OK**.

To Export a Pro/ENGINEER Model to the .ct Neutral Format

Convert a `.ct` file to a CATIA `.model` file with exact solid geometry using the `progeo_to_cat4` utility as follows:

1. In an open part or assembly, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **CATIA to/from File (*.ct)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export CATIA** dialog box opens.
5. If you are exporting an assembly, select **Flat** or **Dittos** from the **File Structure** list to specify the structure of the assembly.
6. To export layers, click **Customize Layers** from the **Export CATIA** dialog box. The **Choose Layers** dialog box opens.
7. Click **Auto ID** to assign layer IDs to layers that do not currently have IDs.
8. Make changes to the **Choose Layers** dialog box and click **OK**.
9. Click **OK** in the **Export CATIA** dialog box to create the intermediate part or assembly `.ct` output file.

To Export a Part or Assembly to a .cat Format File

1. In an open part or assembly, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **CATIA Facets (*.cat)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export CATIA** dialog box opens.
5. If you are exporting an assembly, select **Flat** or **Dittos** from the **File Structure** list to specify the structure of the assembly.
6. To export layers, click **Customize Layers** from the **Export CATIA** dialog box. The **Choose Layers** dialog box opens.
7. Click **Auto ID** to assign layer IDs to layers that do not currently have IDs.
8. Make changes to the **Choose Layers** dialog box and click **OK**.
9. Click **OK** in the **Export CATIA** dialog box. You can convert the `.cat` file to a CATIA `.model` file with a Mock-up solid.

To Convert a .ct or .cat Neutral File to the Native CATIA .model Format

1. Copy the exported CATIA file (`.ct` or `.cat`) and the appropriate utility, `progeo_to_cat4` or `pro_to_cat4`, to a machine with CATIA installed.

2. Run the progeo_to_cat4 utility to convert the .ct file to a .model file with an exact solid, or the pro_to_cat4 utility to convert the .cat file to a .model file with a CATIA Mock-up solid.

   Note: CATIA solids must be available on the receiving side to use progeo_to_cat4.

3. To run interactively, type progeo_to_cat4 or pro_to_cat4. You are prompted for file names and paths.

   If you are in the user input mode, type the name of the .ct or .cat file. The file is converted and placed in the CATIA model file M0000000, unless you specify another name.

Format of the progeo_to_cat4 Utility

The format for using progeo_to_cat4 to convert the .ct file to a .model file is as follows:

```
<progeo_to_cat4> [-n objectname] [-d catia_ddname] [-m
catia_model_path] [-p project] [-i filename] [-v verbose] [-size
index_size data_size]
```

where:

- objectname—Name of the CATIA object.
- catia_ddname—CATIA model number (M1).
- catia_model_path—Path for the CATIA model number.
- project—Name of the CATIA project (default is a blank value consisting of six spaces). If you specify the project name, you automatically specify the catia_ddname.
- filename—Name of the immediate format file to be converted.
- size—Specific index and data table size for the new CATIA model. If you do not specify the size, the default values in the CATIA declaration files are used.

About Configuring the Setup for Direct Import and Export Using Pro/INTERFACE for CATIA

The following utilities are used during data exchange between Pro/ENGINEER and CATIA:

- pro_from_cat4
- progeo_to_cat4

The pro_from_cat4 and progeo_to_cat4 utilities are automatically used during file conversion when you select **CATIA V4 ProGEO Model (*.model)** during import and export from the **Type** list. The CATIA V4 .model files are automatically converted to .ct files during the import and the .ct files are automatically converted to CATIA V4 ProGEO .model files during the export. The system extracts data directly from the CATIA database.

You must have Pro/ENGINEER and CATIA 4.2.0 running on the same SGI, Sun, or Hewlett Packard workstation and run Pro/ENGINEER as a `catusr`.

To Import a CATIA V4 .model File Using Pro/INTERFACE for CATIA

1. Make sure that you have Pro/ENGINEER and CATIA 4.2.0 running on the same workstation.
2. Run Pro/ENGINEER as a `catusr`.
3. Click **File** > **Open**. The **File Open** dialog box opens.
4. Select **CATIA V4 ProGEO Model (*.model)** in the **Type** box. The browser displays the contents of the CATIA Database.
5. Select the CATIA V4 `.model` file you want to import from the list of available files in the CATIA Database.
6. Click **Open**. The **Import New Model** dialog box opens.
7. Select the model type.
8. Type a name for the imported part or assembly or accept the default name in the **Name** box.
9. Click **OK**. The CATIA `.model` files are automatically converted to `.ct` files for import.

To Append a CATIA V4 .model File to a Pro/ENGINEER Part Using Pro/INTERFACE for CATIA

1. Make sure that you have Pro/ENGINEER and CATIA 4.2.0 on the same workstation. The `pro_from_cat4` utility must also be available on the workstation with CATIA installed.
2. Run Pro/ENGINEER as a `catusr`.
3. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
4. Select **CATIA V4 ProGEO Model (*.model)** in the **Type** box. The browser displays the contents of the CATIA Database.
5. Select the CATIA V4 `.model` file you want to append from the list of available files or browse to find the CATIA file.
6. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.
7. If the target model contains coordinate systems, select a coordinate system to locate the geometry of the imported feature or select the default location.
8. If you are appending the file to a Pro/ENGINEER part file with solid geometry, select **Protrusion**, **Cut**, or **Surfaces** to add to the solid geometry, subtract from the solid geometry, or represent the imported geometry as surfaces, leaving the solid geometry unchanged, respectively.
9. Click **OK**.

To Export a Pro/ENGINEER Model to the CATIA V4 ProGEO .model Format Using Pro/INTERFACE for CATIA

1. In an open part or assembly, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **CATIA V4 ProGEO Model (*.model)** in the **Type** box. The contents of the CATIA Database are displayed in the **Save A Copy** dialog box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export CATIA** dialog box opens.
5. Complete the **Export CATIA** dialog box to specify the structure and contents of the output file.

   If you are exporting an assembly, select **Flat** or **Dittos** from the **File Structure** list to specify the structure of the assembly.
6. Click **Customize Layers** from the **Export CATIA** dialog box to export layers. The **Choose Layers** dialog box opens.
7. Click **Auto ID** to manually assign layer IDs to layers that you want to export, but do not currently have IDs.

   You can use the **Choose Layers** dialog box to skip geometry on certain layers.
8. Click **OK** in the **Choose Layers** dialog box.
9. Click **OK** in the **Export CATIA** dialog box. The Pro/ENGINEER model is exported to the `.ct` file and automatically converted to the CATIA V4 ProGEO Model `.model` format. The `.model` file is written to the CATIA Database.

Using Interface for CATIA V5

About Importing CATIA V5 Parts to Pro/ENGINEER

You can import CATIA V5 Revision 6 and later part (`*.CATPart`) files into Pro/ENGINEER with the CATIA V5 interface license option, `intf_for_catia_v5`. You can append the CATPart to an existing Pro/ENGINEER part as an imported feature or to an existing Pro/ENGINEER assembly as a part component. You can import solids, surfaces, and datum entities.

The Interface for CATIA V5 translator supports Associative Topology Bus (ATB). CATIA V5 geometry imported with the `topobus_enable` configuration option set to `yes` can be associatively updated to changes in the native CATIA reference file. For more information on ATB, see the Help on Associative Topology Bus.

To Import a CATIA V5 Part (CATPart) File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **CATIA V5 CATPart (*.CATPart)** in the **Type** box.

3. Select the CATIA V5 part file from the list of available files or browse to find the CATIA V5 file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Type a new import model name in the **Name** box or accept the default.
6. Click **OK**.

   Note: If the version of the CATPart file is earlier than R6, a part with empty import features is created. The Pro/ENGINEER window warns you that empty features are created and asks you to use **Info** > **Geometry Checks** to check the details.

To Append a CATIA V5 Part to a Pro/ENGINEER Part or Assembly

1. Click **Insert** > **Shared Data** > **From File** with a Pro/ENGINEER part or assembly open. The **File Open** dialog box opens.
2. Select **CATIA V5 CATPart (*.CATPart)** in the **Type** box. The CATIA V5 files in the working directory are listed.
3. Select the CATIA V5 file from the list of available files or browse to find the file.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens if you are appending the CATIA V5 file as an imported feature.
   - Accept the default location or select a coordinate system to position the geometry.
   - If the Pro/ENGINEER part contains solid geometry, select **Protrusion**, **Cut**, or **Surfaces** under **Import Data As** to add to the solid geometry, subtract from the solid geometry, or represent the imported geometry as surfaces, leaving the solid geometry unchanged, respectively.
   - Click **OK**. The CATPart is appended to an existing Pro/ENGINEER part as an import feature.
5. If you are appending the CATIA V5 file as a part component to an existing Pro/ENGINEER assembly containing at least one coordinate system, select a coordinate system in the **SEL COORD S** menu.

   Note: If the version of the CATPart file is earlier than R6, a part or part component with empty import features is created. The Pro/ENGINEER window warns you that empty features are created and asks you to use **Info** > **Geometry Checks** to check the details.

To Assemble a CATIA V5 Part in an Existing Assembly

1. Click **Insert** > **Component** > **Assemble** with a Pro/ENGINEER assembly open. The **File Open** dialog box opens.
2. Select **CATIA V5 CATPart (*.CATPart)** in the **Type** box. The CATIA V5 files in the working directory are listed.

3. Select the part file you want to assemble into the existing Pro/ENGINEER assembly as a part component or browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens. It displays the path to the CATIA V5 file with the filename.
5. Type a name for the part component or accept the default name in the **Name** box.
6. Click **OK**. The **Component Placement** dialog box opens.
7. Add placement constraints to position the part or subassembly component.
8. Click **OK**.

   Note: If the version of the CATPart file is earlier than R6, a part component with empty import features is created. The Pro/ENGINEER window warns you that empty features are created and asks you to use **Info** > **Geometry Checks** to check the details.

Importing CATPart Files Earlier Than R6

When you import to Pro/ENGINEER, append to Pro/ENGINEER models, or assemble CATIA V5 part files earlier than revision 6 (R6) into existing Pro/ENGINEER assemblies:

- The resulting import feature in the new or existing Pro/ENGINEER part is empty.
- A warning in the Pro/ENGINEER window states that empty import features are created and prompts you to check the details in the **Info** > **Geometry Checks** > **Troubleshooter** dialog box.
- The **Info** > **Geometry Checks** > **Troubleshooter** dialog box states that the empty import features are created because Pro/ENGINEER does not support the import of CATIA V5 files earlier than R6. The Geom Check is added to the empty import features and you must select an item of the import feature in the **Troubleshooter** dialog box to read the details. You are prompted to save the CATIA V5 files to R6 or higher.
- The import log file in the Information Window shows the version of the imported CATIA file with other details.

Note: Regeneration removes the Geom Check added to the import features.

Perform one of the following tasks:

- Save the CATIA V5 part files to R6 or higher in CATIA. For best results, save to CATIA V5 R8 or higher.
- Delete the empty part, part component, or import feature created by the initial open or insert action.
- Reimport the newly saved version of the CATIA V5 part.

CDRS

About Importing CDRS Models

CDRS is an Associative Topology Bus (ATB)-enabled format. You can import CDRS models or append them to models in Pro/ENGINEER to create IN features that are ATB-enabled.

The following import and append operations create ATB-enabled IN features:

- Opening a CDRS model as a new part with **File** > **Open**.
- Assembling a CDRS model as a part component of an existing Pro/ENGINEER assembly with **Insert** > **Component** > **Assemble**.
- Appending a CDRS model as a new part or part component of an existing Pro/ENGINEER assembly using **Insert** > **Shared Data** > **From File.**
- Appending a CDRS model as a new feature of an existing Pro/ENGINEER part using **Insert** > **Shared Data** > **From File**.

The resulting part file has a standard Pro/ENGINEER part icon in the Model Tree, but the import feature, which is a CDRS IN feature, is represented by an ATB icon.

To Import a CDRS File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **CDRS (*.neu)** as the file type in the **Type** box. Files of the selected type that are located in your working directory are listed.
3. Browse for the file you want if it is not in your working directory.
4. Double-click the name of the part or assembly you want to import. The **Import New Model** dialog box opens.
5. Type a name for the imported part or assembly in the **Name** box or accept the default name. **Part** or **Assembly** is automatically selected.
6. Click **OK**. The file is imported as a new model. A new part with a CDRS IN feature is created.

To Append a CDRS File to a Pro/ENGINEER Model

1. Click **Insert** > **Shared Data** > **From File** in an open part or assembly. The **File Open** dialog box opens.
2. Select **CDRS (*.neu)** as the file type in the **Type** box. Files of the selected type that are located in your working directory are listed.
3. Browse for the file you want if it is not in your working directory.
4. Double-click the file you want to add to your model. If one or more coordinate systems exist, the **Choose Solid Options and Placement** dialog box opens.

5. Accept the default coordinate system in the **Choose Solid Options and Placement** dialog box or use the selection arrow to select another coordinate system.
6. Click **OK** in the **Choose Solid Options and Placement** dialog box.

CGM

About CGM Files

Computer Graphics Metafile (CGM) is an ISO standard that provides a vector-based 2D image file format for the storage and retrieval of graphics information. CGM is a device-independent format.

You can export graphical information to the CGM format in Part, Assembly, and Drawing modes. You can import a CGM file into a drawing, format, layout, or diagram.

Pro/ENGINEER supports CGM versions 1 and 3 for export and versions 1, 2, 3, and 4 for import.

CGM file formats for all versions are in one of the following encodings:

- Binary
- Cleartext

You can import and export graphical information in CGM binary file format. CGM Version 1 supports binary and cleartext encodings.

Both, cleartext or ASCII text and metafile in binary (Milspec), generate files in one of the following units:

- Abstract units (1 unit)
- Metric units (1 inch or 1 cm)

Binary encoding in metric units is the default.

To Export to a CGM File

1. In an open drawing, part, or assembly file, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **CGM (*.cgm)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export CGM** dialog box opens.
5. Select **Abstract Coordinates** or accept **Metric Coordinates**.
6. Select **CLEARTEXT** (ASCII file) or accept **MILSPEC** (binary file) from the **File Structure** list.

7. Click **OK** in the **Export CGM** dialog box.

To Import a CGM File

1. Click **File** > **Open** without a drawing open or click **Insert** > **Shared Data** > **From File** with a drawing open to append a file to a drawing. The **File Open** dialog box opens.
2. Select **CGM (*.cgm)** in the **Type** box.
3. Select the CGM file that you want to import from the list of available files or type the name in the **Name** box.
4. Click **Open**. If you are importing a file with CGM data, the **Import New Model** dialog box opens.
5. Complete the dialog box and click **OK**.

Importing a CGM File

You can import a CGM file in binary and cleartext encoding. Binary encoding in metric units is the default.

When you import a CGM file in metric units, Pro/ENGINEER automatically selects the appropriate standard drawing size.

If the size of the drawing sheet size varies from the standard Pro/ENGINEER drawing sheet size, a drawing sheet of a different and variable size from the standard Pro/ENGINEER drawing sheet size is automatically created in Pro/ENGINEER. This depends on the drawing sheet size information contained in the imported file.

If the size of the drawing sheet matches the standard Pro/ENGINEER drawing sheet size, the drawing is placed on the standard drawing sheet in Pro/ENGINEER.

If you do not specify the drawing sheet size in the file, the drawing is created with the standard Pro/ENGINEER drawing sheet size that is closest to the outline of the imported file.

CGM Entities Supported for Import

The following is a list of entities you can import:

- Appended text
- Cell array if it contains less that 10,000 cells
- Circle
- Circular arc center
- Circular arc center close
- Circular arc 3 point
- Circular arc 3 point close

- Disjoint polyline
- Ellipse
- Elliptical arc
- Elliptical arc close
- Hyperbolic arc
- Non Uniform B-Spline
- Non Uniform Rational B-Spline
- Parabolic arc
- Poly Bezier
- Polygon
- polygon set
- Polyline
- Polymarker
- Rectangle
- Restricted text
- Text

COSMOS Geometry

About COSMOS Geometry

You can use **COSMOS Geom** to create a neutral file that can be read by COSMOS/M (COSMOS/M is the name of a FEM solver from SRAC).

COSMOS/M files are ASCII text files. The system stores them in the current directory. You can rename them and modify them using standard operating system commands. In UNIX, if you create two COSMOS/M files from the same part, the system overwrites the first one you create unless you rename it.

This option is available only in the Part mode.

To Export to a COSMOS File

1. Click **File** > **Save a Copy** in an open part. The **Save a Copy** dialog box opens.
2. Select **Cosmos (*.ntr)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**.

DXF and DWG

Importing DXF and DWG Drawing Files

About Importing DXF or DWG Drawing Files

You can import drawing data from products such as AutoCAD 2002 using the DXF and the DWG file formats, versions 2000 or earlier.

When importing these file formats into Pro/ENGINEER, you can set up and modify configuration options to selectively filter and process data that is specific to the type of entities these file formats support for import. The **Import DXF** and the **Import DWG** dialog boxes are context sensitive and display these configuration options as User Interface options with default values in addition to other import options. You can view and verify the current settings at runtime and revert to original settings, if necessary.

The new and modified configuration option settings are saved in the `current_session.pro` file when you click **OK** on the **Import DXF** or the **Import DWG** dialog box to complete the import of 2D data from these file formats. The **Import DXF** and the **Import DWG** dialog box display the saved values for these options as the default values the next time you open the **Import DXF** or the **Import DWG** dialog box in the same session of Pro/ENGINEER.

You can also set these configuring options using **Tools** > **Options**, but this method does not allow you to easily verify and modify the current settings, make sure if the options are specific to the file type, or revert to the original settings.

You can import DXF files with:

- Blocks as Pro/ENGINEER symbols
- Dimensions as Pro/ENGINEER symbols or as separate entities
- Splines and B-splines
- Xlines as construction lines
- Fonts mapped to fonts in Pro/ENGINEER
- 2D entities and 3D solids where 3D entities are assembled and viewed as assembly components
- Multiple-line text as a single multiple-line note or as multiple single-line notes
- Hatches as draft hatches

  Note: In some instances, tile-like hatches are not imported.

Drawing details, such as views and symbols, especially of assemblies, are structured as blocks in the DXF files. During import, the DXF block structure is maintained and each block is grouped into the corresponding symbol definition. You can use the imported drawing data to create 3D feature-based solid models in Pro/ENGINEER.

If the DXF drawing file consists of multiple drawing sheets of various sizes, you can create a single drawing in Pro/ENGINEER with multiple sheets of variable sizes corresponding to the number of drawing sheets in the DXF file.

You must select more than one layout or drawing sheet for the imported file to create a multiple-sheet drawing in Pro/ENGINEER. You can either import all the sheets contained in the DXF file or selectively import one or more specific sheets of the drawing.

The variable sizes of the drawing sheets are maintained in Pro/ENGINEER. For each sheet of the drawing, a standard-size drawing sheet that is nearest in size to the original drawing sheet is created in Pro/ENGINEER.

Import of DXF and DWG files is also supported on Linux.

To Import a DXF or DWG File as a Drawing

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **DXF (*.dxf)** or **DWG (*.dwg)** in the **Type** box. The DXF or DWG files in the working directory are listed.
3. Select the DXF or DWG file that you want to import or browse to the required file.
4. Click **Open**. The **Import New Model** dialog box opens. **Drawing** is the default.

   Note: Assembly is the default when the DXF or DWG file contains at least a single 3D solid entity. You must explicitly set the model type to **Drawing** to import the 3D solid entities along with the 2D entities in the files.
5. Accept the default file name or type a new name.
6. Click **OK**. The **Import DXF** or **Import DWG** dialog box opens.
7. Select one or more layouts or drawing sheets for the imported drawing from the **Space Name** layout list. If you select more than one layout, the imported file creates a multiple-sheet drawing in Pro/ENGINEER.
8. Under **Import Dimensions**, click one of the following:
   - **As Dimensions**—Imports dimensions as is. This is the default.
   - **As Separate Entities**—Imports the components of dimensions as separate entities.
   - **As Symbols**—Imports dimensions as Pro/ENGINEER symbols.
9. Select the required import options:
   - **Create Associative Dimensions**—Associates the imported dimensions with the corresponding geometry.
   - **Import 3DSOLID entities**—Imports 3D solid entities as views of an assembly.

     Note: Import 3DSOLID entities is available only if the imported DXF file consists of at least a single 3D solid entity.

- **Import Blocks As Symbols**—Imports AutoCAD blocks as symbols in Pro/ENGINEER. For each block instance a separate symbol definition is used.
- **Import Points**—Imports DXF or DWG point entities as drawing points.
- **Create Variable Sheet Size**—Selected by default. Drawing sheets of different sizes, especially for a multiple-sheet drawing with sheets of variable size, are created depending on the information contained in the imported file.
- **Create multi-line text**—Imports the multiple-line text in the imported file as a single multiple-line note.

10. Click **Font Mapping** to map a DXF font to a font in Pro/ENGINEER.
11. Click **OK**. Configuration options corresponding to the options on the **Import DXF** or **Import DWG** dialog box are added with their current values to the `current_session.pro` file.

To Append a DXF or DWG File to a Drawing

1. Open a drawing.
2. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
3. Select **DXF (*.dxf)** or **DWG (*.dwg)** in the **Type** box. The DXF or DWG files in the working directory are listed.
4. Select the file that you want to import from the list of files. The **Import DXF** or **Import DWG** dialog box opens.
5. Select a layout or drawing sheet for the imported drawing from the **Space Name** layout list.

   Note: You cannot select more than one layout for the drawing even if the appended drawing consists of multiple drawing sheets.
6. Under **Import Dimensions**, click one of the following:
   - **As Dimensions**—Imports dimensions as is. This is the default.
   - **As Separate Entities**—Imports dimensions as separate entities.
   - **As Symbols**—Imports dimensions as Pro/ENGINEER symbols.
7. Select the required import options:
   - **Create Associative Dimensions**—Associates the imported dimensions with the corresponding geometry.
   - **Import 3DSOLID entities**—Imports 3D solid entities as views of an assembly.

     Note: Import 3DSOLID entities is available only if the imported DXF file consists of at least a single 3D solid entity.

- **Import Blocks as Symbols**—Imports AutoCAD blocks as symbols in Pro/ENGINEER. For each block instance a separate symbol definition is used.
- **Import Points**—Imports DXF or DWG point entities as drawing points.
- **Create Variable Sheet Size**—Selected by default. Drawing sheets of different sizes, especially for a multiple-sheet drawing with sheets of variable size, are created depending on the information contained in the imported file.
- **Create multi-line text**—Imports the multiple-line text in the imported file as a single multiple-line note.

8. Click **Font Mapping** to map a DXF font to a font in Pro/ENGINEER.
9. Click **OK**. Configuration options corresponding to the options on the **Import DXF** or **Import DWG** dialog box are added with their current values to the `current_session.pro` file.

Import DXF or Import DWG Dialog Box and the Corresponding Configuration Options

The following table lists the User Interface options in the **Import DXF** or the **Import DWG** dialog box that you can configure and set up through the corresponding configuration options. The table also includes the default and other values.

Dialog Box Option	Corresponding Configuration Option	Default Value	Other Values
Import Dimensions	`copy_dxf_dim_pict`	no	`yes, as_symbol`
Create Associative Dimensions	`auto_associate_dimensions`	no	`yes`
Import Blocks as Symbols	`dxf_block_to_pro_symbol`	no	`yes`
Import Points	`intf_in_dwg_pnt_ent`	no	`yes`
Create Variable Sheet Size	`intf_use_variable_size`	yes	`no`
Create multi-line	`intf2d_in_create_multiline_note`	yes	`no`

text			
Import 3DSOLID entities	intf2d_in_acad_ignore_3d	no	yes

Importing a DXF or DWG Drawing File

When you import DXF or DWG data into a drawing:

- Draft entities are created using the units in the drawing setup file. For example, if the drawing setup file sets the units to inches, and the DXF or DWG data is in millimeters, the units are converted to inches during the translation.
- DXF or DWG file values do not override all settings in the drawing setup file. For example, differences can exist in arrow size and style, text size, and parallel or horizontal dimensioning.

  The arrow style of a dimension is stored in the BLOCK SECTION of a DXF file and is treated as part of the dimension picture. Pro/ENGINEER, by default, does not retrieve any data from the dimension picture to recreate the dimension. Therefore, the arrow style of the dimension is lost when you import it.
- AutoCAD entities that are on a blanked layer are transferred to Pro/ENGINEER and placed on a blank layer, after which you can change the display of this layer.
- DXF blocks with symbols, and other details, especially of assemblies, are imported as separate Pro/ENGINEER symbols when you set the `dxf_block_to_pro_symbol` configuration option to `yes`.
- A separate symbol definition is created for each block instance and placed in the appropriate location.

  Specifying different 3D rotations for the block of drafting entities creates several block instances for each block definition. However, the 3D orientation property of the DXF block definition is not maintained. Yet each block instance is placed appropriately so that each block instance reflects its intended appearance when placed and viewed on a drawing sheet in AutoCAD.
- The imported drawing can consist of 2D entities and 3D solids. The 3D solid entities are converted to components of an assembly. You can skip the 3D data in the file to selectively import the 2D data.
- The imported drawing can have more than one layout or drawing sheet and can exist as a multiple-sheet drawing in Pro/ENGINEER. The imported file can have both model space and paper space.
- For a DXF file with multiple drawing sheets of variable size, the size of each drawing sheet in the imported drawing is maintained in Pro/ENGINEER. If you set the `intf_use_variable_size` configuration option to `no`, for each sheet of the drawing, a standard-size drawing sheet that is nearest in size to the original drawing sheet is created in Pro/ENGINEER.

- You can import multiple-line text from a DXF file, with each line of the text in a different font, as multiple single-line notes or as a single multiple-line note. The fonts and styles of the multiple-line text are preserved in the single-line notes.

Importing Layers

DXF and DWG file formats support the export and import of layers. Pro/ENGINEER places the entities imported from DXF or DWG on a layer in the Drawing Mode. The layer ID of an imported entity in Pro/ENGINEER is the same as its layer ID in the DXF or DWG file.

Exporting Drawings to DXF and DWG

About Exporting Drawings to DXF or DWG Files

You can export drawing data from Pro/ENGINEER to products such as AutoCAD 2002 using file versions 2000 and earlier of the formats:

- DXF
- DWG

Version 2000 of the DXF and the DWG file formats is the default. Export to the DWG and DXF file formats is also supported on Linux.

To export a drawing, drawing format, or layout to a DXF or DWG file, you must create a DXF or DWG file for every sheet of the drawing.

The **Export Environment for DXF** and the **Export Environment for DWG** dialog boxes display options specific to the respective file types and the type of entities these file types support for export. These options depend on the additional configuration options that you set and are saved automatically when you click **OK** or **Export** in the **Export Environment for DXF** or the **Export Environment for DWG** dialog box to complete the export to the IGES format.

You can view and verify the current settings at runtime while saving or exporting the DXF or DWG drawing file. You can modify the configuration options and their values while saving a copy of the drawing.

The log file created during the export has details about the export such as the file type version, the configuration options settings, the assembly structure representation as blocks or layers, export of text as text or geometric entities, and so on. You can view and verify the information, edit, and save the log file in an Information Window after the export.

A Pro/ENGINEER:

- Spline or B-spline is exported to the DXF format as a spline.
- Construction line is exported to the DXF format as an XLINE.
- Draft hatch is exported to the DXF format as hatch. Pro/ENGINEER draft hatches create AutoCAD UDF hatches.

- Multiple-line note is exported to the DXF and DWG file formats as a single MTEXT entity or as multiple TEXT entities.

DWG and DXF Blocks

Using the `intf_out_layer` configuration option, you can export user-defined symbols and drawing groups through DXF and DWG as blocks, making it is easier to modify the exported drawings. Two-dimensional representations of individual components with their drafting entities for each view within an assembly are also translated as blocks.

To Export to a Drawing, Drawing Format, or Layout File

1. Click **File** > **Save a Copy** in a drawing. The **Save a Copy** dialog box opens.
2. Select **DXF (*.dxf)** or **DWG (*.dwg)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. Accept the default name or type a new name for the model in the **New Name** box.
4. Click **OK**. The **Export Environment for DXF** or the **Export Environment for DWG** dialog box opens.
5. Select a version for the file type. You can select DXF or DWG versions 12, 13, or 14. DXF or DWG file version 2000 is the default.
6. Select the option specific to the entity-type that you want to export. You can export the following entities:
    - Splines as is or as polylines
    - Hatches as is or as separate entities
    - Points as is or as shapes
    - Notes as text or as strokes or separate geometric entities
    - Multiline notes as MText
    - Text alignment as is or as fit
    - Assemblies as layers or blocks
    - Comment lines
    - Dimensions lines with breaks on witness lines to separate entities and dimensions as geometric entities
7. Click **Open Log Window**. The Information Window displays the Export Log File with the details of the export.
    - Click **Edit** to edit the Export Log File.
    - Click **File** > **Save** to save the file.

- Click **Close**.

8. Click **OK** in the **Export Environment for DXF** or the **Export Environment for DWG** dialog box to export the file and quit the dialog box or click **Export** to export the file without closing the dialog box.

   The settings in the **Export Environment for DXF** or the **Export Environment for DWG** dialog box are saved automatically when you click **OK** or **Export**.

   The drawing is exported to the DXF or the DWG format and a file with one of the following names is created:

   - `<drawingname>.dxf` or `<drawingname>_#.dxf`.
   - `<drawingname>.dwg` or `<drawingname>_#.dwg`.

     where `#` is the sheet number for a multisheet drawing.

Controlling DXF and DWG Export

Make sure that you have set the following configuration options:

- `use_export_2d_dialog` to `yes` so that the **Export Environment for DXF** or **Export Environment for DWG** dialog box opens.
- `dxf_export_format` or `dwg_export_format` to `12`, `13`, `14`, or `2000`.
- `intf2d_out_enhanced_ents` to control the export of polylines, lines, and arcs, or hatch and spline entities.
- `intf2d_out_pnt_ent` to export points as is or as shapes.
- `dxf_out_stroke_text yes` to export text as strokes in DXF or DWG files.
- `intf_out_layer` as:
  - `block_layer`
  - `part_layer`
  - `none`
- `dxf_out_comments` to `yes` to create comment lines in the DXF file.
- `intf2d_out_acad_mtext` to `yes` to convert the multiple-line text notes in the Pro/ENGINEER drawing to a single MTEXT entity in the DXF or DWG file.
- `intf2d_out_acad_text_align` to `as_is` or `fit` to align text.
- `dxf_out_sep_dim_w_breaks` to `yes` to export dimensions with breaks on witness lines so that entities are separated and the original picture is preserved.
- `intf2d_out_open_log_window` to `yes` to display the window with the Export Log File.

Setting intf_out_layer to block_layer

`Use intf_out_layer block_layer` to export components (parts), symbols, groups (for each view), and tables for Pro/ENGINEER drawings as AutoCAD blocks. Blocks are translated as follows:

- User-defined symbols are translated as a named blocks. Instances of the symbol, are translated as separate blocks. The default block name is `<Symbolname>_<Instname>`.
- User-defined groups created in Drawing and Diagram are translated as a named blocks. The default block name is `<Symbolname>_<Groupname>`.
- Tables with associated text created in a Pro/ENGINEER drawing are translated as a named block. The default block name is `table_<instance_name>`.
- Default block names for individual components are

  `<part_name>_<view_name>_<instance_name>`

Setting intf_out_layer to part_layer

Use `intf_out_layer part_layer` to export data from Pro/ENGINEER drawings to DXF and DWG format that can:

- Create pre-defined layers.
- Map Pro/ENGINEER line styles to AutoCAD line styles.
- Get the exported drawing in a 1:1 scale regardless of the original scale.
- When exporting, put in blocks for each hatching and table that is included in the drawing.
- Update the text export. Text is exported as AutoCAD text and not within an outside block.

Layering Scheme

You need to create a layering scheme for each part you are exporting to DXF or DWG. If a part is found more than one time, its entities go to the relevant same single layer.

DXF/DWG Layer Name	Entities
`<part_name>_Dxf_Axis`	All axes
`<part_name>_Dxf_Continuous_Line`	All continuous lines
`<part_name>_Dxf_Hidden_Line`	All hidden lines
`<part_name>_Dxf_Dimension`	All dimensions
`<part_name>_Dxf_Text`	All text

<part_name>_Dxf_Hatching	All hatching
<part_name>_Dxf_Table	All tables
<part_name>_Dxf_Balloon	All balloons
<part_name>_Dxf_Format	All format entities

Setting intf_out_layer to none

DXF and DWG support the export and import of layers. When you export layers containing Pro/ENGINEER entities to DXF and DWG and intf_out_layer is set to none:

- Layers are exported with their original name
- An entity belonging to more than one Pro/ENGINEER layer is exported with its lowest alphabetical layer name. For example, an entity on layer apple and layer pear exports on layer apple.

The following table shows how the system assigns entities to layers in a DXF or DWG file created by Pro/ENGINEER and exported to DXF or DWG.

Font Used by an Entity	Layer Assigned
Centerline	DEFAULT_1
Hidden line	DEFAULT_2
Phantom line	DEFAULT_3

If an entity has been assigned to a layer but it does not fall into any of the preceding categories, it will go to that layer. If an entity does not belong to any layer, it goes to a default zero layer.

Mapping Layers and Line Styles

Map layers and line styles in the dxf_export.pro file. To map layers exported from Pro/ENGINEER to DXF and DWG, use the dxf_export.pro file. The syntax for layer mapping is:

```
map_layer <items to map> <Ext)
```

For example if the dxf_export.pro file contains the line map_layer Dxf_Axis 5, and you are exporting part1, the resulting layer is part1_5. This layer contains all the axes that belong to part1 and are found in the exported drawing.

Note: If a particular type of entity does not exist in the part, then that layer is not created. This applies to newly created layers only.

To map line styles, use the following syntax in the dxf_export.pro file:

```
map_line_style <Pro/E name> <AutoCAD name>
```

For example: map_line_style CTRLFONT CENTER

Note: A 32-character limit exists for DXF file names. Therefore, a layer name with more than 32 characters is truncated.

Text, Hatching, and Tables

Text is exported as AutoCAD text, but not within a block. Multiple-line notes are exported as a single MTEXT entity or as multiple TEXT entities.

All hatching is converted to AutoCAD blocks. Each hatching group is converted to a single block and placed according to the layering scheme.

Each table is converted to a single AutoCAD block and according to the layering scheme.

Export Drawing Scale

When you export a Pro/ENGINEER drawing to the DXF or DWG format, Pro/ENGINEER outputs the drawing scale to the `DIMFLAC` DXF/DWG environment variable. The export of the scale information is controlled by the `dxf_out_drawing_scale` configuration option.

When `dxf_out_drawing_scale` is set to `no` (the default), the drawing is exported without scale information.

When `dxf_out_drawing_scale` is set to `yes`

- Scale information exports correctly for non-scaled views.
- Detailed and other non-scaled views export the correct picture but do not transfer scaled dimensions.
- When exporting multiple models with different scales, only the non-scaled views of the current model export with the correct dimension factor.

View Scale

When exporting, the entire drawing is rescaled so the main view scale becomes 1:1. For example, if the main view scale is 2:1, the drawing size is scaled .5, and all views including detail views, are scaled .5. A detail view scale of 4:1, for example, becomes 2:1.

All draft entities including the drawing format are resized at a scale factor of .5. For example, if the text within the dimension or notes had an original height of .125, it is resized to a height of .0625. In AutoCAD, it appears to be the same as the drawing that exists in Pro/ENGINEER. When an entity is measured in one of the standard views, the result is the same as the dimension.

Color Mapping Pro/ENGINEER Defaults to AutoCAD

You can map Pro/ENGINEER system colors to DXF and DWG standard colors using the `dxf_export.pro` configuration file. This file must be located in the Pro/ENGINEER startup directory To map Pro/ENGINEER default system colors to similar or equivalent AutoCAD system colors, set up ther `dxf_export.pro` file as follows:

```
map_color BACKGROUND_COLOR 186
map_color DIMMED_COLOR 9
map_color LETTER_COLOR 2
map_color HIGHLIGHT_COLOR 1
map_color EDGE_HIGHLIGHT_COLOR 5
map_color GEOMETRY_COLOR
map_color HIDDEN_COLOR
map_color SHEETMETAL _COLOR
map_color CURVE_COLOR
map_color VOLUME_COLOR
map_color SECTION_COLOR
```

Default Pro/ENGINEER System Colors

The default system colors for Pro/ENGINEER are:

Pro/ENGINEER System Color Name	Pro/ENGINEER Default Color
LETTER_COLOR	yellow
HIGHLIGHT_COLOR	red
GEOMETRY_COLOR	white
DIMMED_MENU_COLOR	light gray
EDGE_HIGHLIGHT_COLOR	blue
HIDDEN_COLOR	dark gray
VOLUME_COLOR	magenta
SECTION_COLOR	cyan
SHEETMETAL_COLOR	green
CURVE_COLOR	brown
BACKGROUND_COLOR	Dark blue

The background color does not convey the actual screen color to AutoCAD. This color is used in color manipulation by Pro/ENGINEER.

Basic AutoCAD System Colors

The basic AutoCAD colors to which you can map are as listed.

AutoCAD Color(s)	AutoCAD Number(s)
Red	1
Yellow	2
Green	3
Light blue	4
Dark blue	5
Magenta	6
Black	7
Gray	8
Light gray	9
Gray shades	250-255
Full color palette	10-249

Example: Line in dxf_export.pro File

If you want to export all Pro/ENGINEER geometry to green (AutoCAD color 3), include the following line in the `dxf_export.pro` file:

```
map_color GEOMETRY_COLOR 3
```

Importing DXF Files with 3D Data

About Importing DXF Files with 3D Data

You can import into Pro/ENGINEER parts, assemblies, components, and features created in AutoCAD and saved as 3D DXF files. The 3D DXF and DWG files can consist of both 2D and 3D data.

3D data is of the following types:

- Faceted
- Embedded exact ACIS data
- 3D solid entities

You can import DXF files containing:

- Faceted geometry as features of existing parts, new parts or assemblies, or as part or assembly components of existing assemblies.
- Embedded ACIS data only as a new assembly or an assembly component.

- A combination of 2D entities and 3D solids as drawings. The 3D solids are assembled and viewed as components of an assembly after import.

Importing Faceted And Embedded ACIS Data

The default model type in the **Import New Model** dialog box is **Drawing** when the AutoCAD 3D DXF files contain faceted and embedded ACIS data. To import the faceted data, you must explicitly set the model type to **Part** or **Assembly** in the **Import New Model** dialog box. Embedded ACIS geometry can only be imported as a new assembly or an assembly component. Use the imported 3D geometry for constructing features and creating design models.

Importing 2D entities with 3D Solid Entities

The default model type is set to **Assembly** in the **Import New Model** dialog box when the AutoCAD 3D DXF or DWG file contains at least a single 3D solid entity. You must explicitly set the model type to **Drawing** in the **Import New Model** dialog box to import the 3D solid entities along with the 2D entities in the files. Entities defined as ACIS geometry (BODY, REGION) are ignored.

Importing DXF or DWG Files with 3D Solids and 2D Entities as Drawings

When you import a 3D DXF or DWG file that contains 3D solids and 2D entities as a drawing,

- The 3D solids in the DXF or DWG file create the 3D model in session. The 3D solids are assembled as components of an assembly.
- The 2D draft entities in the DXF or DWG file create the 2D drawing in session.
- The 3D solids and the 2D entities are placed in model and paper space, respectively.
- 3D views are created when the 3D solid entities belong to a viewport.

  A viewport entity in DXF defines the projection of a model-space object in paper space.
- A viewport is imported as a 2D view when no 3D solids are attached to the viewport.
- The 2D entities that belong to the same viewport as the 3D entities are attached to the 3D views.
- If the DXF or DWG file does not contain 2D data, 3D views are created without the 2D entities attached to them. If the file consists of only 2D data, only 2D views are created in Pro/ENGINEER.
- The 3D model is associated with the 2D drawing and this association is maintained when you build on the 3D model or modify the drawing.

You can skip the solid entities and selectively import the 2D contents using the `intf2d_in_acad_ignore_3d` configuration option.

Note: The `intf2d_in_acad_ignore_3d` configuration option is available as the **Import 3DSOLID entities** option on the **Import DXF** and the **Import DWG** dialog boxes when the 3D DXF or the DWG file contains at least a single solid entity. This option allows you to imports 3D solid entities as views of an assembly.

Importing AutoCAD 3D DXF Files as Parts

When you import an AutoCAD 3D DXF file as a part with **Part** selected in the **Import New Model** dialog box:

- A single faceted solid is imported as a new part with a faceted solid.
- Multiple faceted solids are also imported as a new part with multiple faceted solids on separate layers. The imported faceted solids merge into a single faceted entity in the new part.

  If the faceted solids exist as one solid per layer, a flat assembly with one faceted component per faceted solid is created.
- 3D faces, polygonal mesh, and standard polylines are imported as faceted data, that is, surfaces or solids. Polylines are imported as wireframe entities. The imported feature consists of faceted and wireframe entities.
- If the DXF file contains only standard polylines, an import feature with wireframe entities is created.
- If the DXF file contains embedded exact ACIS data, the embedded exact ACIS data is skipped.
- If the DXF file contains faceted data, the faceted data is imported.

When you import a feature created in AutoCAD and saved as a DXF file:

- The feature is appended to an existing part as a faceted import feature.
- The embedded exact ACIS data is skipped.

Importing AutoCAD 3D DXF Files as Assemblies or Assembly Components

When you import the AutoCAD 3D DXF file as an assembly with **Assembly** selected in the **Import New Model** dialog box, embedded ACIS data and the faceted data are imported as separate components with faceted data or exact data.

If the DXF file does not contain embedded ACIS geometry, the assembly created in Pro/ENGINEER consists of multiple faceted part components with a flat structure. If a single layer of the DXF file contains multiple faceted solids, a subassembly representing the layer is created.

If the DXF file contains multiple packets of embedded ACIS data along with faceted data, a component for each packet of embedded ACIS data and separate components for the faceted data are created.

When you import a component

- The component is appended to an existing assembly as a faceted component of the assembly.

- Embedded ACIS data is appended to an existing assembly as a subassembly component with separate components for the faceted data and the exact ACIS data.
- A part component is appended to an existing assembly as a faceted part component. The ACIS data is skipped.
- A subassembly component is appended to an existing assembly as a subassembly component with faceted components. The embedded ACIS data creates separate subassembly components.

Filtering 3D Geometric Data

A 3D DXF file containing faceted solids with a single solid on each layer is imported as a flat assembly with a single faceted component for each solid.

A 3D DXF file containing faceted solids with multiple solids on each layer is imported as an assembly with subassemblies representing the layers containing a part component for each solid on that layer.

You can use the **Layer Import Options** dialog box to filter faceted data or the faceted solids.

If faceted solids and embedded ACIS data of the imported assembly exist on separate layers, you can set the import status of the layers with faceted data to skip to filter faceted data and import only embedded ACIS data.

To Import an AutoCAD 3D DXF File

1. Click **File** > **Open** without a part or an assembly open. The **File Open** dialog box opens.
2. Select **DXF (*.dxf)** in the **Type** box. The DXF files in the working directory appear.
3. Select the DXF file you want to import or browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens. The default import type is **Drawing**.

   Note: Assembly is the default when the DXF file contains at least a single 3D solid entity.
5. Select **Part** or **Assembly** to import the faceted or embedded ACIS geometry.

   If the DXF file contains embedded ACIS geometry,you can only import the file as a new assembly or an assembly component.

   Note: Do not import the DXF file with the embedded ACIS geometry as a feature or a new part.
6. Accept the default name for the part or the assembly or specify a new name.
7. Click **OK**.

- If you select **Part** as the import type, the faceted data merge to form a single faceted representation in a single import feature.
- If you select **Assembly** as the import type, the faceted data form part components of subassemblies represented by layers.

To Append an AutoCAD 3D DXF File to an Existing Part or Assembly

1. Click **Insert** > **Shared Data** > **From File** with a part or assembly open.

   Click **Insert** > **Component** > **Assemble** to assemble a DXF file with 3D data into an existing assembly.

   The **File Open** dialog box opens.
2. Select **DXF (*.dxf)** in the **Type** box. The DXF files in the working directory appear.
3. Select the DXF file you want to append to an existing part or assembly. Browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens. The default import type is **Drawing**.
5. Select **Part** or **Assembly**.
6. Accept the default name for the part or the assembly or type a new name.
7. Click **OK**. The **Choose Solid Options and Placement** dialog box opens.

   The **Component Placement** dialog box opens if you are assembling the file into an existing assembly.
8. Accept the default coordinate system or select a coordinate system to position the appended geometry.

Granite-based Files

About Directly Opening Granite-based Files in Pro/ENGINEER

The method of directly opening Granite-based models, such as Pro/DESKTOP `.des` parts and assemblies, as new models in Pro/ENGINEER is the default. For each feature in the Granite-based model, a corresponding read-only feature is created in the directly opened model in Pro/ENGINEER. You cannot redefine the features created in the model or subsequently regenerate the directly-opened model. You can only reference the imported geometry to create features and components in the model.

For a directly opened Granite-based file, the feature-structure of the Granite file is preserved in the resultant Pro/ENGINEER model though these features are read-only in Pro/ENGINEER. The imported feature-types are represented by their corresponding icons in the Model Tree. The original item and feature IDs and configurations that correspond to the family table instances in Pro/ENGINEER are

preserved in the model. Nongeometric information such as layers, colors, and parameters are imported. Sketches, datums, and patterns are not supported.

You can append or assemble the directly opened models as part or subassembly components of existing assemblies in Pro/ENGINEER.

ICEM

About Importing ICEM Surf Models

ICEM is an Associative Topology Bus (ATB)-enabled data exchange format. ICEM surfaces, when imported into Pro/ENGINEER or appended to models in Pro/ENGINEER, create ATB-enabled import IN features.

The following import and append operations create ATB-enabled IN features:

- Opening an ICEM model as a new part with **File** > **Open**.
- Assembling an ICEM model as a part component of an existing Pro/ENGINEER assembly with **Insert** > **Component** > **Assemble**.
- Appending an ICEM model as a new part or part component of an existing Pro/ENGINEER assembly using **Insert** > **Shared Data** > **From File.**
- Appending an ICEM model as a new feature of an existing Pro/ENGINEER part using **Insert** > **Shared Data** > **From File**.

ATB allows the associative update of imported data when changes are made to the native ICEM data.

To Import an ICEM File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **ICEM (*.icm)** as the file type in the **Type** box. Files of the selected type that are located in your working directory are listed.
3. Browse for the file you want if it is not in your working directory.
4. Double-click the name of the part or assembly you want to import. The **Import New Model** dialog box opens.
5. Type a name for the imported part or assembly in the **Name** box or accept the default name. **Part** or **Assembly** is automatically selected, indicating the type of entity you have selected to import.
6. Click **OK**. The file is imported as a new model.

To Append an ICEM File to a Pro/ENGINEER Model

1. Click **Insert** > **Shared Data** > **From File** in an open part or assembly. The **File Open** dialog box opens.

2. Select **ICEM (*.icm)** as the file type in the **Type** box. Files of the selected type that are located in your working directory are listed.
3. Browse for the file you want if it is not in your working directory.
4. Double-click the `.icm` file you want to add to your model. If one or more coordinate systems exist, the **Choose Solid Options and Placement** dialog box opens.
5. Accept the default coordinate system in the **Choose Solid Options and Placement** dialog box or use the selection arrow to select another coordinate system.
6. Click **OK** in the **Choose Solid Options and Placement** dialog box.

I-DEAS

About Importing I-DEAS Model and Package Files to Pro/ENGINEER

You can import I-DEAS file types `.mf1` (model) and `.pkg` (package). The `.mf1` and `.pkg` files are packages of multiple parts and assemblies. You can install I-DEAS and Pro/ENGINEER on separate workstations. Install Pro/ENGINEER on a server machine or the local workstation. Install I-DEAS on a local workstation or run it from a server machine. However, they must be installed on the same platform type. The `.mf1` model files can only be imported from within projects of the I-DEAS Project Database and not from any other location. The `.pkg` files can be imported from the working directory or any other location.

You cannot directly open I-DEAS parts and assemblies as individual files in Pro/ENGINEER. Instead, you can filter a selected `.mf1` or `.pkg` file during import to pick parts and assemblies to include in the import. The models are imported as components of a top-level assembly, but this assembly is only a 'package' necessary for the import process. You can discard this top-level assembly after import is complete and you have saved the 'component' models.

You cannot append or assemble a `.mf1` or a `.pkg` file to an existing Pro/ENGINEER part or assembly. You can only access this import file through **File** > **Open**. You cannot redefine the reference file of an imported feature and select a `.mf1` or a `.pkg` file as the replacement file.

Configuring the I-DEAS Interface

You must ensure the following for the import of I-DEAS files to Pro/ENGINEER:

- I-DEAS installation has the `OpenIdeas` license.
- On UNIX, the following `OpenIdeas` libraries exist in the library-path environment variable where `<ideas_loadpoint>` specifies the location of the I-DEAS installation:
    - `<ideas_loadpoint>/oirun/lib`

    - <ideas_loadpoint>/orbix/shlib
    - <ideas_loadpoint>/orbix/shlib/default
- For Windows, include <ideas_loadpoint>\oirun\lib and <ideas_loadpoint>\Iona\bin.
- Use the configuration option, intf3d_ideas_install_dir, to specify the full path to the I-DEAS loadpoint.
- The following are the platform-specific library-path environment variables:
    - SHLIB_PATH for HP
    - LD_LIBRARY_PATH for Solaris
    - LD_LIBRARYN32_PATH for SGI
    - Path for Windows
- Ensure that I-DEAS is installed and the ORBIX daemon is running.
- Ensure that you start Pro/ENGINEER from a directory that does not include a space in its name or path.
- Use the configuration option, intf3d_ideas_install_dir, to specify the full path to the I-DEAS loadpoint.

During the import, if the I-DEAS startup command is other than ideas, the Pro/ENGINEER message window states that the I-DEAS startup command is not in the system path and cannot be located. Stop the import process and set the value of the intf3d_ideas_run_command configuration option to ideas.

To Import an I-DEAS Model (*.mf1) File

1. Set the intf3d_ideas_install_dir configuration option to accurately point to the I-DEAS installation.

   Note: Ensure that the intf3d_ideas_run_command configuration option is set to the I-DEAS startup command, that is the value, ideas.

2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **I-DEAS Model (*.mf1)** in the **Type** box.
    - The I-DEAS Project Database is visible in the **Look In** box.
    - The I-DEAS project folders are displayed.
    - I-DEAS part and assembly files have the same .prt and .asm file extensions as Pro/ENGINEER part and assembly models. They are displayed as Pro/ENGINEER files in the **Open** dialog box when the Pro/ENGINEER part or assembly type filters are set if they conform to the Pro/ENGINEER file-naming conventions. However, you cannot open them in Pro/ENGINEER.

   Note: You can only import .mf1 files from one of the virtual project folders. You cannot import a .mf1 file found outside the I-DEAS Project Database. To import

I-DEAS .mf1 files from within the projects of the I-DEAS Project Database, the part and assembly models of .mf1 files must be submitted to the model file bin.

4. Double-click an I-DEAS project folder and select a .mf1 file.
5. Click **Open**. The **Import New Model** dialog box opens. It shows the name of the .mf1 model file
6. Type a name for the imported file or accept the default name in the **Name** box.
7. Click **OK**. The **Layer Import Options** dialog box opens if you have set the intf3d_ideas_import_filter configuration option to yes.
   - The **Layer Import Options** dialog box lists the part and assembly models contained in the .mf1 file.
   - The import status of all the models is **Show**, by default.
8. To filter models that you do not require, set their import status to **Skip**.
9. Click **OK** on the **Layer Import Options** dialog box. Only models with their import status set to **Show** are imported.

   The selected part and assembly models are imported as components of a top-level Pro/ENGINEER assembly. You can discard this top-level assembly because it is only a 'package' that is required for the import of the individual part and assembly models contained in the .mf1 file.

   Note: Part models of the .mf1 file not put into the model file bin are excluded in the imported assembly.

To Import an I-DEAS Package (*.pkg) File

1. Set the intf3d_ideas_install_dir configuration option to accurately point to the I-DEAS installation.

   Note: Ensure that the intf3d_ideas_run_command configuration option is set to the I-DEAS startup command, that is the value, ideas.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **I-DEAS Package (*.pkg)** in the **Type** box. By default, a list of .pkg files in the working directory is displayed.

   Note: I-DEAS part and assembly files have the same .prt and .asm file extensions as Pro/ENGINEER part and assembly models. These I-DEAS .prt and .asm files are colocated with .pkg files. They are displayed as Pro/ENGINEER files in the **Open** dialog box when the Pro/ENGINEER part or assembly type filters are set if they conform to the Pro/ENGINEER file-naming conventions. However, you cannot open them in Pro/ENGINEER.
4. Select a .pkg file from the working directory or browse to select a .pkg file from any other directory.

Note: If a part or assembly is missing in the `.pkg` file, an error message in the Pro/ENGINEER window states that the import has failed and that the data may be corrupt or incomplete.

5. Click **Open**. The **Import New Model** dialog box opens with the file path and name.
6. Type a name for the imported file or accept the default name in the **Name** box.
7. Click **OK**. The **Layer Import Options** dialog box opens if you have set the `intf3d_ideas_import_filter` configuration option to `yes`.
   - The **Layer Import Options** dialog box lists the part and assembly models contained in the `.pkg` file.
   - The import status of all the models is **Show**, by default.
8. To filter models that you do not require, set their import status to **Skip**.
9. Click **OK** on the **Layer Import Options** dialog box. Only models with their import status set to **Show** are imported.

   The part and assembly models are imported as components of a top-level Pro/ENGINEER assembly. You can discard this top-level assembly because it is only a 'package' that is required for the import of the individual part and assembly models contained in the `.pkg` file.

Troubleshooting When Importing from I-DEAS

If the import process terminates abnormally, and you see the `IDEAS2PROE_*.mf1` temporary model files on disk, ask your I-DEAS administrator to delete the `IDEAS2PROE_*.mf1` files and the `_IDEAS2PROE_*` libraries under the `_IDEAS2PROE_` project to free disk space.

If the import process is abruptly interrupted, check the `error*.out` file in the Pro/ENGINEER working directory for details.

If the Pro/ENGINEER message window displays the error that an I-DEAS project or a library in the I-DEAS project cannot be created, contact your I-DEAS administrator to make sure that you have the package import permissions.

I-DEAS `*.pkg` files are platform-dependent, that is, a file generated on Windows cannot be imported into Pro/ENGINEER on UNIX, and vice versa. If the Pro/ENGINEER message window states that the `*.pkg` file selected for import to Pro/ENGINEER is not compliant on your platform, convert the `*.pkg` file to the platform-specific format using the I-DEAS converter available at `<ideas_loadpoint>/bin/hdsconvert`.

If you find that the `IdeasToPro` process is running even after the import has failed or Pro/ENGINEER has terminated abnormally, kill the `IdeasToPro` process manually, start Pro/ENGINEER if required, and import the I-DEAS file again.

IGES

Importing IGES Files to Drawing Mode

About Importing IGES Data Into a Drawing

The Initial Graphics Exchange Specification (IGES) transfers graphical and textual information between computer aided design systems. When reading IGES data into a drawing, note the following:

- You can set up and modify configuration options to selectively filter and process data that is specific to the type of entities that the IGES file format supports for import. The **Import IGES** dialog box is context sensitive and displays these configuration options as User Interface options with default values in addition to other import options. You can view and verify the current settings at runtime and revert to original settings, if necessary. These configuration option settings are saved to the `current_session.pro` file when you complete the import of 2D data from the IGES file. The **Import IGES** dialog box displays the saved values for these options as the default values the next time you open **Import IGES** in the same session of Pro/ENGINEER.
- When the imported drawing sheets vary in size, drawing sheets of different sizes are created depending on the information contained in the imported file.
- If the drawing sheet size matches the standard Pro/ENGINEER drawing sheet size, the drawing is placed on the standard size drawing sheet format in Pro/ENGINEER.
- If you do not specify the drawing sheet size in the file, the drawing is placed on a standard Pro/ENGINEER drawing sheet size that is closest to the outline of the imported file.
- The lower-left corner of the drawing is the origin of draft entities in the drawing. When the system reads an IGES file into the drawing, it places the file relative to the drawing origin.
- Draft entities are created using the units specified in the drawing setup file. For example, if the drawing setup file sets the units to inches, and the units for the IGES data is in millimeters, the units of the IGES entities are converted to inches during the translation. For example, 25.4 mm in the IGES file becomes 1 inch in the drawing.
- Multiple-line text is imported as a single multiple-line note or as multiple single-line notes with the original fonts.
- Typically, the IGES file values do not override the settings of the drawing setup options. Arrow size and style, text size, and parallel/horizontal dimensioning are examples of areas where differences can occur.

- When you import drawings, some line styles might appear different. For example, when you import drawings from CATIA, dots can look like dashes, and spaces can appear larger.
- When you choose **Modify** and **Value** for an imported dimension, you are placed in Modify, Text, and Line modes to edit the selected text line of the dimension. IGES does not represent the value of a dimension as a parameter, but only as graphics. Thus, the displayed text contains an "@O" (oh) that is inserted to set the origin of the line of the text.

Diagrams, Drawings, Formats, Layouts, and Reports

You can import IGES files containing geometry and annotation into a diagram, 2D drawing, format, layout, or report. IGES files with 3D model entities use the view definitions to place the geometry in the drawing. The resulting entities are 2D draft entities.

When you import IGES files that contain geometry and annotations:

- You can place IGES groups on separate layers on import. You must set the `iges_in_group_to_dwg_layer` configuration option to `yes`. The IGES groups are converted to Pro/ENGINEER layers with the name `IGES_GRP_<line number of the IGES group>`.
- You can import a series of IGES files into Pro/ENGINEER using the `proigsutil` utility.
- The proper drawing format is automatically selected if the format of the imported IGES drawing corresponds to a standard size (portrait or landscape) drawing.

To Import a 2D IGES File in Drawing Mode

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. In the **Type** box, select **IGES (.igs, .iges)**. The IGES files in the current working directory are listed.
3. Click the name of the 2D IGES file that you want to import or browse to the required IGES file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Click **Drawing**. The **Import IGES** dialog box opens.
6. Under **VIEWS**, click one of the following:
    - **No Views**—Does not import views from the IGES file.
    - **2D Views**—Imports the views in the IGES file as 2D views.
    - **3D Views**—Creates 3D views in the imported IGES file. Available only when a 3D model exists in the IGES file.
7. Select the required import options:

- **Create Associative Dimensions**—Associates the imported IGES dimensions with the corresponding imported geometry.
- **Import Points**—Converts the IGES point entities to drawing points in Pro/ENGINEER.
- **Create Variable Sheet Size**—Selected by default. Drawing sheets of different sizes are created, depending on the information contained in the imported file when the imported drawing sheets vary in size.
- **Import User Colors**—Imports the color-definition entities in the IGES file as user-defined colors.
- **Import User Line Fonts**—Imports user-defined line fonts.
- **Import Groups As Layers**—Converts an IGES group to layers in the drawing.
- **Create multi-line text**—Imports the multiple-line text in the imported file as a single multiple-line note.

8. Click **OK**. Configuration options corresponding to the options on the **Import IGES** dialog box are added with their current values to the `current_session.pro` file.

To Append an IGES Drawing File to a Pro/ENGINEER Drawing

1. Open a diagram, drawing, format, layout, or report file.
2. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
3. In the **Type** box, select **IGES (.igs, .iges)**. The IGES files in the current working directory are listed.
4. Click the name of the 2D IGES file that you want to append to the existing Pro/ENGINEER drawing or browse to the required IGES file.
5. Click **Open** to append the IGES file to the diagram, drawing, format, layout, or report file that is open. The **Import IGES** dialog box opens.
6. Under **VIEWS**, click one of the following:
   - **No Views**—Does not import views from the IGES file.
   - **2D Views**—Imports the views in the IGES file as 2D views.
   - **3D Views**—Creates 3D views in the imported IGES file. Available only when a 3D model exists in the IGES file.
7. Select the required import options:
   - **Create Associative Dimensions**—Associates the imported IGES dimensions with the corresponding imported geometry.
   - **Import Points**—Converts the IGES point entities to drawing points in Pro/ENGINEER.

- **Create Variable Sheet Size**—Selected by default. Drawing sheets of different sizes are created, depending on the information contained in the imported file when the imported drawing sheets vary in size.
- **Import User Colors**—Imports the color-definition entities in the IGES file as user-defined colors.
- **Import User Line Fonts**—Imports user-defined line fonts.
- **Import Groups As Layers**—Converts an IGES group to layers in the drawing.
- **Create multi-line text**—Imports the multiple-line text in the imported file as a single multiple-line note.

8. Click **OK**. Configuration options corresponding to the options on the **Import IGES** dialog box are added with their current values to the `current_session.pro` file.

Importing IGES View Layout

When you create a new Pro/ENGINEER drawing by importing a 2D IGES file containing 3D view and 3D model information, Pro/ENGINEER creates a model and a drawing that uses the model to display the relevant views. This is the case when you are creating a new model (**File** > **Open**) only, not when you are appending an existing model (**Insert** > **Shared Data** > **From File**). The following occurs when you import a 2D IGES file into a drawing:

- If IGES views do not exist in the 2D IGES model that you are importing, a drawing without 2D or 3D views is created.
- If IGES views exist but there is no IGES 3D model definition, a drawing with 2D views is created.
- If IGES views and an IGES 3D model definition exist, a model and a drawing with the relevant 3D views is created.

Configuration File Options Supporting the Import of IGES Files

The following table lists the configuration options that support the import of IGES files into drawings, formats, layouts, parts, and assemblies.

Import Option	Drawing	Part	Assembly
`edge_display_quality`	yes	yes	yes
`explode_iges_dimension_note`	yes		
`auto_associate_dimensions`	yes		
`fix_autocad_iges_text_scale`	yes		
`fix_boundaries_on_import`	yes		

fix_catia_iges_sym_note	yes		
fix_imported_set_view_orient	yes		
iges_clip_view_note	yes		
iges_in_dwg_color	yes		
iges_in_assoc_dim_geom_21	yes		
intf_in_dwg_view	yes		
iges_in_dwg_line_font	yes		
iges_in_dwg_pnt_ent	yes		
iges_in_group_to_dwg_layer	yes		
iges_note_disp	yes		
iges_zero_view_disp	yes		
recompute_iges_dim_value	yes		
intf_in_blanked_entities		yes	yes
iges_in_106_f2_as_spline		yes	yes
intf3d_in_close_open_boundaries		yes	yes
intf3d_in_include_items		yes	yes
intf_in_extract_profiles			
intf_in_treat_polyline_as		yes	yes
intf2d_in_create_multiline_note	yes		
intf_use_variable_size	yes		

2D IGES Entities Supported for Import into Drawing Mode

The following table lists the 2D IGES entities that you can import into Drawing mode.

IGES Entity Type	IGES Entity Name	IGES Entity Form	Configuration File Option Controlling Import
100	Circular Arc		
102	Composite		

	Curve		
104	Conic Arc		
106	Copius Data	11, 12 Polylines 20– Centerline through points 21– Centerline through circle centers –31-38– Section 40–Witness Line 63–Simple Closed Planar Curve	
108	Plane		
110	Line		
112	Parametric Spline Curve		
116	Point		`iges_in_dwg_pnt_ent`
124	Transformation Matrix		
126	Rational B-spline Curve		
202	Angular Dimension		
206	Diameter Dimension		
210	General Label		
212	General Note		
214	Leader (Arrow)		
216	Linear Dimension		

218	Ordinate Dimension		
222	Radius Dimension		
228	General Symbol		`iges_out_symbol_entity`
230	Sectioned Area		
302	Associativity Definition		
308	Subfigure Definition		
304	Line Font Definition		`iges_in_dwg_line_font`
314	Color Definition		`iges_in_dwg_color`
320	Network Subfigure Definition		
402	Associativity Instance	3–Views visible 4–Views Visible, Color, Line Weight 13–Dimensioned Geometry 21–Dimensioned Geometry (replacing form 13)	
404	Drawing		
406	Property Entity	15–Name 16–Drawing Size 17–Drawing Units	
		Drawing Sheet Size	`intf_use_variable_size`
408	Singular Subfigure		

	Instance		
410	View		

Importing Colors in Drawing Mode

You can import colors in drawing, part, and assembly modes.

You can use the `iges_in_dwg_color` configuration option to set the color number of all entities with user-defined colors in an IGES file to point to the corresponding color definition entity. If you set this option to `no` (the default), the color definition entities in the IGES file are ignored when you import the drawing. Every entity referencing these colors are set to use the color assigned in the color definition entity. If you set this option to `yes`, the color definition entities in the IGES file are imported to the drawing as user-defined colors, and all entities are set to use them.

Importing 3D IGES Files

To Import a 3D IGES File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **IGES (.igs, .iges)** in the **Type** box. A list of IGES files in the current working directory appear.
3. Click the name of the 3D IGES file that you want to import or browse to find and then click the file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Click **OK**. The `IGES <filename>.log` file appears in the **INFORMATION WINDOW**. If there are layers associated with the IGES part or assembly, and you have set the `intf_in_layer_asm_dialog` configuration option to `yes`, the **Layer Import Options** dialog box opens.
6. Close or minimize the **INFORMATION WINDOW**.
7. In the **Layer Import Options** dialog box, make changes as necessary.
8. Click **OK**.

To Append a 3D IGES File to an Existing Model

1. Click **Insert** > **Shared Data** > **From File** with a file open. The **File Open** dialog box opens.
2. Select **IGES (.igs, .iges)** in the **Type** box. A list of IGES files in the current working directory appear.
3. Click the name of the 3D IGES file that you want to append to the existing Pro/ENGINEER model or browse to find and then click the file.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.

5. Select:
   - **Protrusion** to create a solid protrusion from the imported geometry and add it to the existing part geometry.
   - **Cut** to create a solid cut in the existing geometry, using the imported geometry.
   - **Surfaces** to represent the imported geometry as a collection of quilts. Adjacent surfaces will be joined whenever possible.
6. Click **OK** in the **Choose Solid Options and Placement** dialog box.

Entities Supported for Import into Drawing, Part, and Assembly Modes

The following table lists the entities you can import into Drawing, Part, and Assembly modes.

IGES Type	IGES Name	IGES Form	Reference	Result in Pro/ENGINEER
100	Circular Arc		Independent	An arc entity
			Ref. by 142	An edge of a surface
			Ref. by 102	An arc in a composite curve
102	Composite Curve		Independent	A composite curve
			Ref. by 142	Edges of a surface
104	Conic Arc		Independent	A spline curve
			Ref. by 142	An edge of a surface
			Ref. by 102	A spline in a composite curve
106	Copious Data	2	Independent	If `iges_in_106_f2_as_spline` is set to `no`–point entities `yes`–spline
		11	Ref. by 142	Edges of a surface
		12	Independent	Composite curve of linear segments

		63	Ref. by 142	Edges of a surface
108	Plane	1		Planar surface with an external boundary
		-1		Internal boundary of a planar surface (a hole in a plane)
		0	Independent	Datum plane
110	Line		Independent	A line entity
			Ref. by 142	An edge of a surface
			Ref. by 102	A component of a composite curve
112	Parametric Spline		Independent	A spline curve(s)
			Independent	A spline curve(s)
			Ref. by 142	An edge or edges of a surface
			Ref. by 102	A spline in a composite curve
114	Spline Surfaces		Independent	A cubic spline surface(s)
			Ref. by 144	Trimmed spline surface(s)
116	Point		Independent	A point entity
118	Ruled Surface		Independent	A ruled surface entity
			Ref. by 144	Trimmed rules surface(s)
120	Revolved Surface		Independent	A revolved surface entity
			Ref. by 144	Trimmed revolved surface(s)
122	Tabulated Cylinder		Independent	A tabulated cylinder surface
			Ref. by 144	Trimmed tabulated cylinder surface

124	Transformation	0, 1	Ref. by *	Entity * is transformed
126	B-Spline Curve		Independent	A spline curve or curves
			Ref. by 142	An edge of a surface
			Ref. by 102	A spline or splines in a C-curve
128	Rational B-Spline Surface		Independent	Spline surface or surfaces
			Ref. by 144	Trimmed spline surfaces
141	Boundary		Ref. by 143	Edges of trimmed surface(s)
142	Curve on Surface		Ref. by 144	Edges of trimmed surface(s)
143	Bounded Surface		Independent	A trimmed surface(s)
144	Trimmed Surface		Independent	A trimmed surface(s)
208	Flag Note Entity		Independent	Note with a flag
314	Color Definition		Independent	You can color (according to that definition) curves and surfaces that reference this entity.
402	Associativity Instance	1, 7, 14, 15	Independent	You can put IGES Group entities on a separate layer upon import in a Pro/ENGINEER drawing.
		7		Creates a layer in a part or assembly.

		21	Independent	If the configuration file option `iges_in_assoc_dim_geom_21` and the drawing setup file option `associative_dimensioning` are set to `yes`, supports import of dimensioned geometry for radial, diameter, and ordinate dimensions. If these options are set to `no`, imported dimensions are nonassociative.
410	View		Independent	A named view of a model

Importing Colors in Part and Assembly Modes

You can import colors in drawing, part, and assembly modes.

You can import colors of IGES curves and surfaces, both standard (red, green, blue, yellow, magenta, cyan) and user-defined. When the system imports them, it puts the colored surfaces and curves on named layers. The naming convention for layers is as follows:

- **For standard colors**—`INTF_<COLOR>`, where `<COLOR>` is one of the following: RED, GREEN, BLUE, YELLOW, MAGENTA, CYAN.
- **For user-defined colors (IGES Entity 314, Color)**—If the name of the color is present, the layer name corresponds to the color name (for example, BLUE). If the color name is unspecified, the layer name is `INTFCLR_<XXX>`, where `<XXX>` is the IGES ID of the color entity.

About Importing IGES Files in Sketcher Mode

You can import IGES files containing 2D curve geometry into Sketcher mode.

Most IGES files do not contain dimensions, or if they do, there is no associativity between the dimensions and the geometry. Therefore, you need to dimension most imported IGES files in Sketcher mode. If the associativity between dimensions and geometry is established, Pro/ENGINEER attempts to use this dimensioning scheme.

Note: When importing an IGES file, Pro/ENGINEER assumes that the IGES file contains only 2D entities in the xy-plane, and does not check for errors. Sketcher mode does not support subfigures and subfigure instances.

To Import an IGES File in Sketcher Mode

1. Click **File** > **New**. The **New** dialog box opens.

2. Click **Sketch**.
3. Click **OK** to open a file in Sketcher mode.
4. Click **Sketch** > **Data from File**. The **File Open** dialog box opens.
5. Select the IGES file you want to import from the list of available files.
6. Click **Open** to create the IGES file.
7. After the system displays the 2D entities in your Sketcher mode window, you can add or delete entities and dimensions.

Exporting from Drawing Mode to IGES

About Exporting to IGES

Before you export data to IGES files, determine which IGES entities the receiving system can receive, then use the appropriate configuration file options to export that data type.

The **Export Environment for IGES** dialog box opens if the `use_export_2d_dialog` configuration option is set to the default value `yes`. The options on the **Export Environment for IGES** dialog box depend on the additional configuration options that you have set. These option settings are saved automatically when you export the drawing to the IGES format.

You can view and verify the current settings during runtime while saving or exporting the drawing file. You can modify the configuration options and their values while saving a copy of the drawing.

The log file created during the export has details about the export such as the IGES version, the configuration options and their values, and so on. You can view and verify the information, edit, and save the log file after the export.

The log file is created in the same directory as the target `*.iges` file and is named:

- `igesout.log.#`—For a drawing file
- `<name>_out.log.#`—For a part or assembly file

You can create multiple IGES files in a single session using Pro/BATCH.

Note: IGES does not support all of the special characters that Pro/ENGINEER supports. Therefore, when you export to the IGES file, some characters are not visible. When you export special symbols that are not supported by IGES, the receiving system draws these symbols in strokes. For example, clipped dimensions with two arrowheads are exported with only one arrowhead.

You must create an IGES file for every sheet of the drawing.

IGES Version in Output Files

The header of an IGES file contains the code for IGES version 5.2. If the receiving system does not accept 5.2, you can edit the header so that it contains the most

recent version of IGES the receiving system accepts (for example, 4.0). The receiving system can then read all of the file.

Pro/ENGINEER Entity Attributes that IGES Supports

The following table lists the Pro/ENGINEER entity attributes that IGES supports and the corresponding modes.

Attribute	Mode
Thickness	Supported for export of compressed shell model in FEM mesh export.
Line Font	Supported for Drawing mode only; ignored for models.
Blank status	Supported for Part and Assembly modes. The `intf_out_blanked_entities` and `intf_in_blanked_entities` configuration options switch the import and export of entities with blank status.
Line weight	Supported for Drawing mode only.
Color	Import of color from an IGES file is supported for the following IGES codes: IGES Code 8, Color White, Pro/ENGINEER Color Geometry IGES Code 5, Color Yellow, Pro/ENGINEER Color Leader IGES Codes 2, 6, Color Red, Pro/ENGINEER Color Highlight IGES Codes 4, 7, Color Blue, Pro/ENGINEER Color Edge Highlight
Cross hatch	Supported in Drawing mode only.

To Export a Drawing to an IGES Drawing File

1. Set the following configuration options:
   - `iges_out_symbol_entity`
   - `intf2d_iges_out_hatch`
   - `intf2d_out_pnt_ent`
   - `iges_export_dwg_views`
   - `iges_out_dwg_line_font`
   - `iges_out_dwg_color`
2. Click **File** > **Save a Copy** in a drawing. The **Save a Copy** dialog box opens.

3. Select **IGES (.igs)** in the **Type** box.
4. Accept the default name in the **New Name** box or type a new model name for the export.
5. Click **OK**. The **Export Environment for IGES** dialog box opens.
6. Select the options specific to the type of entity you want to export. You can export the following entities:
    - Splines as is or as polylines
    - Cross hatches as separate single geometric entities or as IGES Element Type 230.

      If you set the `intf2d_iges_out_hatch` configuration option to `yes`, IGES Element Type 230, that is, a sectioned area, is created for each cross hatch.
    - Symbols as is or as separated entities.
    - Points as is or as shapes.

      Note: By specifying the interface quality, you can determine if an IGES drawing file contains overlapping entities.
    - Views
    - Line fonts as is or as solid lines
    - Colors as is or as white
7. Click **Open Log Window**. The Information Window displays the log file with details of the export.
8. In the Information Window, click **Edit** to edit the log file.
9. Click **File** > **Save** to save the file.
10. Click **Close**.
11. Click **OK** in the **Export Environment for IGES** dialog box to export the file and quit the dialog box or click **Export** to export the file without closing the dialog box.
    - The settings in the **Export Environment for IGES** dialog box are automatically saved.
    - The drawing is exported and a file named `<drawingname>.igs` or `<drawingname>_#.igs` is created, where `#` is the sheet number for a multi-sheet drawing.

IGES Entities Supported for Export from Drawing Mode

The following table lists the 2D IGES entities that you can export in Drawing mode.

IGES Entity Type	IGES Entity Name	Configuration File Option Controlling Export
100	Circular Arc	
106	Copius Data Form: 11–Polylines 31–Section 40–Witness Line 63–Simple Closed Planar Curve	iges_out_ent_as_bspline
108	Clipping Planes	
110	Line	
116	Point	intf2d_out_pnt_ent
124	Transformation Matrix	
202	Angular Dimension	
206	Diameter Dimension	
210	General Label	
212	General Note	
214	Leader (Arrow)	
216	Linear Dimension	
218	Ordinate Dimension	
222	Radius Dimension	
228	General Symbol	iges_out_symbol_entity
230	Sectioned Area Cross hatches	intf2d_iges_out_hatch
304	Line Font Definition	iges_out_dwg_line_font
314	Color Definition	iges_out_dwg_color
404	Drawing	
406	Property Entity Form:	iges_export_dwg_views

	15–Name 16–Drawing Size 17–Drawing Units –Views	
410	View entities	

Enabling CADRA Export

In Drawing mode, you can enable CADRA export by adding the option `enable_cadra_export` to the configuration file. This adds the **CADRA** option to the **EXPORT** menu, enabling you to create a CADRA-specific IGES file.

Note: You can export layers containing draft entities in Drawing mode.

To Export Colors in Drawing Mode

You can export colors in drawing mode.

If you set the `iges_out_dwg_color` configuration option to `no` (the default), the user-defined colors in the drawing are ignored when you export an IGES file, and sets every entity referencing these colors to use white color in the IGES file. If you set this option to `yes`, the user-defined colors in the drawing are exported to the IGES file as color definition entities, and all entities using them are given a pointer to the corresponding color definition entity in the IGES file.

Colors Not Supported

This option supports the definition of all user-defined colors, except the following:

- Geometry (white, IGES color code 8)
- Leader (yellow, IGES color code 5)
- Highlight (red, IGES color code 2, 6)
- Edge Highlight (blue, IGES color code 4, 7)

Exporting IGES Groups

About IGES Groups

An IGES group is a collection of drawing notes and model edges that belong to a specific drawing view and component, and are associated within the receiving system.

Use `iges_export_dwg_views` configuration option to replace the IGES group. If you set this configuration option to `yes`, all entities related to a specific view receive the same view ID in the IGES file.

Pro/ENGINEER exports IGES groups as IGES associativity entity type 402, form 7. Although IGES does not assign a particular associativity to these groups, some receiving systems support them and assign a special meaning, such as features or

components, to them. If you have a Pro/DETAIL license, you can create IGES groups.

Pro/ENGINEER exports IGES grouped view entities (2D views only) for the following entity types: 108, 404, 406:15, 406:16, 406:17, and 410. If you have a Pro/LEGACY license and use the `intf_in_dwg_view` configuration option set to `2D_VIEWS`, you can use 2D views to manipulate the view by moving the view or changing the scale. Dimensions created on these views understand the view scale and create the correct dimension. Entities that are sheet annotations are referenced to IGES overlay view.

When selecting items for an IGES group, keep in mind the following:

- For a part, you must select model edges, cosmetic features, and silhouette edges from the same member in the same view. If you select one member of a cosmetic feature, you select all members of this feature.
- For an assembly, all entities must belong to the same assembly member shown in the same view.
- You can use standard notes and balloon notes from any view or those that are unattached.
- You can add layers to a group.

To Create an IGES Group

1. In a drawing, click **Tools** > **IGES Group**. The **IGES GROUPS** menu appears.
2. Click **Create**.
3. Type a name for the IGES group, then select a drawing view in a part drawing or an assembly member in an assembly drawing from which you want to select items.
4. Select one of the following from the **SEL OPTION** menu:
    - **Sel Items**—Select items by selecting from the screen. As you select items, they are highlighted.
    - **Sel Layers**—Select items by choosing a layer to which they belong. Choose the layer name from the **LAYER SEL** menu. An IGES group can contain more than one layer.
    - **Sel Chain**—Creates a group consisting of a chain of edges by specifying the first and the last element in the chain. To switch between the chain edges, use **Next** and **Previous** in the **CHOOSE** menu. To finish, choose **Accept**.
5. After you have made your selections, click **Done Sel**.

To Edit an IGES Group

1. In a drawing, click **Tools** > **IGES Group**. The **IGES GROUPS** menu appears.
2. Click **Edit**.
3. Type the name of an existing group and click checkmark, or press ENTER to quit.

4. Select one of the following from the **SEL OPTION** menu:
    - **Sel Items**—Adds items to the group or removes them from the group. Choose one of these options from the **EDIT IGES GP** menu:

      Add Items—Adds selected items to the current group.

      Remove Items—Removes selected items from the current group.
    - **Sel Layers**—Changes layers in the group. Removes a check mark from the layers you want to remove, and checks those you want to add to the current IGES group.
    - **Sel Chain**—Adds a chain of edges by specifying the first and the last element in the chain.
5. To confirm your selections, click **Done Sel**.

To Delete an IGES Group

1. Click **Tools** > **IGES Group**. The **IGES GROUPS** menu appears.
2. Click **Delete**.
3. Type the name of an existing group to delete and click checkmark, or press ENTER to quit.

To List an IGES Group

In the current drawing, select **List** from the **IGES GROUPS** menu. The system displays an **Information Window**, listing existing groups.

Exporting Parts and Assemblies to IGES

To Export a Part or Assembly to an IGES File

1. In an assembly or part click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **IGES (*.igs)** in the **Type** list.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export IGES** dialog box opens.
5. Complete the dialog box to specify the structure and contents of the output file.
6. If you want to export layers, click **Customize Layers**. The **Choose Layers** dialog box opens.
7. Click **Auto ID** to assign layer IDs to layers that do not have IDs.
8. Make changes to the **Choose Layers** dialog box, then click **OK** to accept the changes made.

9. Click **OK** to create the part or assembly IGES file.

Part and Assembly IGES Files

Assembly IGES Files

When exporting assembly files to IGES, the system generates an IGES file with the suffix `_asm` appended to the name of the file. This is to prevent overwriting a part file with an assembly file of the same name, as illustrated.

In Pro/ENGINEER	In IGES Format
`desk.prt`	`desk.igs`
`desk.asm`	`desk_asm.igs`

Part IGES Files

You can tailor part IGES files to the requirements of the receiving system. For example, you need not give surface information to a system that can only process curve geometry.

IGES Entities Supported for Export from Part and Assembly Modes

The following table lists the IGES entities that you can export from Part and Assembly modes.

Pro/ENGINEER Entity	Type	IGES Entity and Name
Circular edge	100	Circular Arc
Composite curves	102	Composite Curve
Planar surface	114	Spline Surface, B-Spline Surface if `iges_out_spl_srfs_as_128` is set to `no`
	128	B-Spline Surface if `iges_out_spl_srfs_as_128` is set to `yes` (default)
Spline curve or edge	112	Parametric Spline Curve if `iges_out_spl_crvs_as_126` is set to `no`
	126	B-Spline Curve if `iges_out_spl_crvs_as_126` is set to `yes` (default)
Linear edge	110	Line

Spline surface (datum or solid)	114	Parametric Spline Surface if `iges_out_spl_srfs_as_128` is set to `no`
	128	B-Spline Surface if `iges_out_spl_srfs_as_128` is set to `yes` (default)
Blended or swept surface (datum or solid)	114	Spline Surface if `iges_out_spl_srfs_as_128` is set to `no`
	128	B-Spline Surface if `iges_out_spl_srfs_as_128` is set to `yes` (default)
Revolved surface (datum or solid)	120	Surface of Revolution
Extruded spline surface (datum or solid)	122	Tabulated Cylinder
	124	Transformation Matrix
Spline curve or edge	126	B-Spline Curve
Spline surface (datum or solid)	128	B-Spline Surface
Datum or solid surface	141	Boundary if `iges_out_trm_srfs_as_143` is set to `yes` (default is `no`)
Datum or solid surface	142	Curve on Surface
Datum or solid surface	143	Bounded surface if `iges_out_trm_srfs_as_143` is set to `yes` (default is `no`)
Datum or solid surface	144	Trimmed Surface
Supports export of assemblies in One Level and All Level modes	308	Singular Subfigure Definition
Thickness attribute of a compressed shell model	406	Form 11

A list of files of assembly components	406	External Reference File (form 12)
Supports export of assemblies in One Level and All Level modes	408	Singular Subfigure Instance
Supports export of assemblies in One Level and All Level modes	416	External Reference

File Structure for Exporting Assemblies to IGES

You can tailor part IGES files to the requirements of the receiving system. For example, you need not give surface information to a system that can only process curve geometry.

When you export an assembly to IGES, you can specify the structure and contents of the output files. The following options are found in the **File Structure** list in the assembly **Export IGES** dialog box:

- **Flat**—Exports all of the geometry of the assembly to a single IGES file as if it were a part. When retrieved into another system, the assembly acts as one part. You should place each part on a layer to differentiate between parts in the receiving system.
- **One Level**—Exports an assembly IGES file with external references pointing to the IGES files of its components. This file contains only top-level geometry (for example, assembly features).
- **All Levels**—Exports an assembly file with external references to all components as well as all the component IGES files. It creates component parts and subassemblies with their respective geometry and external references. This option supports all levels of hierarchy.
- **All Parts**—Exports an assembly to IGES as multiple files containing geometry information of its components and assembly features, if any. These parts use the same reference coordinate system to ease reassembly in the receiving system. Specifically, the system creates the following IGES files:
    - An assembly IGES file with the name `<modelname>_asm.igs`, where `<modelname>` is the name of the assembly containing any assembly feature geometry.
    - A subassembly IGES file for each subassembly (if any) with the name `<modelname>_cpy_#.igs`, where `<modelname>` is the name of the subassembly, and `#` identifies the occurrence of the same subassembly in the assembly containing any assembly feature geometry.

- Part files of all component parts. If a component appears more than once in the assembly, for each occurrence of the component there is a corresponding IGES part file. The naming convention for part files is <partname>_cpy_#.igs, where <partname> is the name of the part, and # identifies the occurrence of the same part in the assembly. The system creates multiple part files, since each instance is in a different position relative to the reference coordinate system.

Using Proigsutil

About Using proigsutil

You can use proigsutil, a Pro/ENGINEER IGES conversion utility to convert a list of IGES files to Pro/ENGINEER drawings or parts. This utility converts only IGES files.

Use the following command to execute proigsutil:

```
proigsutil <pro_command>
```

Follow it with these arguments:

```
-help -ok -s src_dir -l filename -r -t target_dir -k mins -tmp
```

If you execute proigsutil without arguments, the utility runs in interactive mode and you are prompted for the necessary information.

Command Mode

The following table describes the argument values.

Argument Values	Description
-help	Displays this information.
-if	Import format that indicates whether to read IGES file information as a Pro/ENGINEER part or drawing. Default: none (you must specify drawing or part.)
-ie	The IGES file extension. Default: igs
-ok	Confirms that you are satisfied with the backup of the files to be converted. Default: If flag is not present, system prompts you for an answer.

-s	Name of a directory containing files to convert (the starting directory). Default: The current directory.
-l	Name of a file containing a list of file names to convert. File names (for example, box.prt) should be listed one per line. Do not specify a version; the system converts the latest files. Default: Converts all of the files in the starting directory.
-r	Recursively converts files in directories encountered below the starting directory. Default: Converts only the files in the starting directory.
-t	Name of a directory in which to write converted files. Default: The starting directory.
-k	The maximum time (in minutes) allowed for the conversion of a file. After this time, the system aborts the process converting the current file. Default: 0 (that is, no limit).
-tmp	Temporary directory that proigsutil uses. The directory partition must have at least enough space for the storage of the largest file to be converted. Default: /tmp.
-ms	The minimum disk space (Kb) required to do conversions. The system checks this after each conversion; proigsutil is aborted if there is not enough space. Default: 0 (that is, no limit).
-c	One of the following: <filename>.pro–Uses the specified configuration file before each conversion. existing (default)–Uses the configuration file in the conversion directory (if available). none–Does not use a configuration file, even if one exists in the conversion directory.

The following are command mode examples:

- `proigsutil pro`

  Converts all files in this directory and accepts all defaults except backup status.

- `proigsutil pro -if drw -ok`

  Converts all `.igs` files in this directory to Pro/ENGINEER drawings.

- `proigsutil pro -if prt -r -ok`

  Converts all `.igs` files in this tree to Pro/ENGINEER parts.

Example: Converting Drawings in Interactive Mode Using proigsutil

Type: `proigsutil`

```
proigsutil v10.0—the Pro/ENGINEER conversion utility
-----------------------------------------------------
Interactive mode—you will be prompted for answers, hit <RETURN>
to get default values in ()s when available.
Use 'proigsutil -help' for more information, including command-line
arguments.

Name of command to run Pro/ENGINEER?: pro

Convert to parts or drawings (choice: drw prt)

[drw] ?: <CR>

Import file extension

[igs] ?: <CR>

Name of directory containing files to convert

[/home/users/user_2/drawings/iges] ?: <CR>

Name of file containing list of files to convert

hit <RETURN> to convert all files ?: draw_iges.txt

Name of directory in which to put converted files

[/home/users/user_2/drawings/iges] ?:
/home/users/user_2/drawings/convert

Maximum time for any file conversion (choice: number of mins, 0 =
unlimited)

[0] ?: 20

Temporary work directory

[/tmp] ?: <CR>

Minimum disk space for any conversion (choice: size in Kb, 0 =
unlimited)

[0] ?: <CR>
```

```
Name of a config. file to use (choice: none, existing OR a filename)
[existing] ?: <CR>
Satisfied with the backup of the files to be converted (choice: yes no)
[no] ?: yes
WARNING: Maximum file conversion time (20 mins) is less than 1 hour.
This may be too short a time.
1st pass—estimate conversion space requirements
-------------------------------------------------
Total size of files to convert: 91 (Kb)
Total estimated size of converted files: 91 (Kb)
2nd pass—Process the files (in /tmp/10289/list.txt)
-------------------------------------------------
--- CHECKING THE CONFIG FILE ---
[1] 10641
[1] + Killed /tmp/10289/killer 60
CONFIG FILE OK.
PROCESSING FILE 1 OF 2 - /home/users/user_2/drawings/iges/cam_draw.igs
PROCESSING FILE 2 OF 2 -
/home/users/user_2/drawings/iges/check_detail.igs
```

Example: Converting Parts in Interactive Mode Using proigsutil

```
Type: [proigsutil]
proigsutil v10.0—the Pro/ENGINEER conversion utility
-------------------------------------------------
Interactive mode—you will be prompted for answers, hit <RETURN>
to get default values in []’s when available.
Use ‘proigsutil -help’ for more information, including command-line
arguments.
Name of command to run Pro/ENGINEER ?: pro
Convert to parts or drawings (choice: drw prt)
[drw] ?: prt
Import file extension
[igs] ?: <CR>
Name of directory containing files to convert
[/home/users/user_2/models/part_iges] ?: <CR>
Name of file containing list of files to convert
```

```
hit <RETURN> to convert all files ?: <CR>
Recursively convert files in dirs encountered (choice: yes no)
[no] ?: <CR>
Name of directory in which to put converted files
[/home/users/doug/part_iges_files] ?: /home/users/user_2/convert
Store geometry when converted (choice: geom no geom)
[no geom] ?: <CR>
Maximum time for any file conversion (choice: number of mins, 0 =
unlimited)
[0] ?: <CR>
Temporary work directory
[/tmp] ?: <CR>
Minimum disk space for any conversion (choice: size in Kb, 0 =
unlimited)
[0] ?: <CR>
Name of a config. file to use (choice: none, existing OR a filename)
[existing] ?: <CR>
Satisfied with the backup of the files to be converted (choice: yes no)
[no] ?: yes
1st pass—estimate conversion space requirements
-------------------------------------------------
Total size of files to convert: 178 (Kb)
Total estimated size of converted files: 178 (Kb)
2nd pass—Process the files (in /tmp/11101/list.txt)
-------------------------------------------------
PROCESSING FILE 1 OF 10 - /home/users/user_2/models/part_iges/base_2.igs
PROCESSING FILE 2 OF 10 - /home/users/user_2/models/part_iges/block.igs
PROCESSING FILE 3 OF 10 - /home/users/user_2/models/part_iges/bolt.igs
PROCESSING FILE 4 OF 10 -
/home/users/user_2/models/part_iges/control_arm.igs
PROCESSING FILE 5 OF 10 - /home/users/user_2/models/part_iges/cover.igs
PROCESSING FILE 6 OF 10 -
/home/users/user_2/models/part_iges/cylinder.igs
PROCESSING FILE 7 OF 10 - /home/users/user_2/models/part_iges/slide.igs
PROCESSING FILE 8 OF 10 - /home/users/user_2/models/part_iges/clamp.igs
```

```
PROCESSING FILE 9 OF 10 - /home/users/user_2/models/part_iges/pin.igs

PROCESSING FILE 10 OF 10 -
/home/users/user_2/models/part_iges/socket.igs
```

MEDUSA

About Exporting and Importing Parts, Assemblies, and Drawings to MEDUSA

You can export Pro/DETAIL drawings to MEDUSA from within Pro/ENGINEER. Pro/DETAIL drawings are exported as MEDUSA sheets.

Note: The exchange of drawing data between Pro/ENGINEER and MEDUSA is not supported on Linux.

You can also export solid model information about parts and assemblies to MEDUSA. Geometry is exported to MEDUSA as tessellated data. MEDUSA exports to an ASCII-based `.asc` file for visualization purposes.

The coordinate systems of the Pro/ENGINEER model that exist on visible layers are exported to MEDUSA with the tessellated geometry. If the coordinate systems lie on blanked layers, they are not exported and are ignored during the export. Use the exported coordinate systems in the MEDUSA Plant Design (MPD) tool for routed systems references.

You can import tessellated `.asc` MEDUSA files as faceted data.

To Export a Part or Assembly to MEDUSA

1. Click **File** > **Save a Copy** in an active part or assembly. The **Save a Copy** dialog box opens.
2. Select **Medusa (*.asc)** in the **Type** box. The **Export Medusa** dialog box opens.
3. Change the values for the maximum **Chord Height** and **Angle Control** if you want additional control of tessellation.

   For assemblies only, select:

   - **All Parts** to export the entire assembly.
   - **Include** to select each part you want to include in the file.
   - **Exclude** to select each part you want to exclude from the file.

   The message line below these three options indicates the number of parts you are exporting. It is updated each time you exit the **GET SELECT** menu. Switching between **Include** and **Exclude** does not affect your selections.

   To cancel your selections, click **Reset**.
4. To select or create a coordinate system that defines the xyz space, click **GET COORDS** by clicking the selection icon below **Coordinate System**. You can use

the default coordinate system of the model, but this can result in non-positive coordinates.

5. Type the name of the file without the extension or accept the default name.
6. Fix the edge tessellation problems if problem surfaces are found.
7. Click **Apply** to export the part or assembly or click **OK** to export and close the dialog box.

To Import a MEDUSA File of a Part or Assembly

1. Choose **File** > **Open**. The **File Open** dialog box opens.
2. Click **Medusa (*.asc)** from the **Type** list. The MEDUSA files in the working directory appear.
3. Select the MEDUSA file that you want to import from the list of available files or browse to find the file.
4. Accept the default name for the MEDUSA part or assembly you want to import or type a new name in the **Name** box.
5. Click **Open** to complete the import of the MEDUSA file. The resulting Pro/ENGINEER model is a part or an assembly depending on the contents of the file.

To Append a MEDUSA File to a Pro/ENGINEER Part or Assembly

1. Click **Insert** > **Shared Data** > **From File** with a part or an assembly open.

   Click **Insert** > **Component** > **Assemble** to assemble a MEDUSA file as a new part or subassembly component of an assembly.

   The **File Open** dialog box opens.
2. Select **Medusa (*.asc)** in the **Type** list.
3. Click the file you want to append to an existing part or assembly from the list of files.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.
5. Accept the default coordinate system or select a coordinate system. The imported MEDUSA `.asc` file appends to the Pro/ENGINEER part or assembly depending on the contents of the file.

Pro/DETAIL to MEDUSA

To Export from Pro/DETAIL to MEDUSA

1. With a drawing active, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.

2. Select **Medusa (s.*)** in the **Type** box.
3. Accept the default export file name or type a new name.
4. Click **OK** to create the MEDUSA file.

Mapping Pro/DETAIL Information to MEDUSA

The following occurs when you export a Pro/DETAIL sheet to MEDUSA:

- Exported Pro/ENGINEER grouped entities create MEDUSA clumps. Grouped entities include hatching, tables, user-defined symbols, groups, and part and assembly components.
- Entity layer information is mapped.
- Parametric notes are mapped to MEDUSA text with their attributes.
- The model name is mapped to a sheet attribute.

Importing MEDUSA Drawings

To Import a MEDUSA File with Drawing Data

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **Medusa (s.*, *.she)** in the **Type** box. MEDUSA files in the working directory appear.
3. Select the MEDUSA file you want to import or browse to find the file.
4. Click **Open**. The **Import New Model** dialog box opens. **Drawing** is the default.
5. Accept the default file name or type a new name.
6. Click **OK**.

To Append a MEDUSA Drawing File to a Pro/ENGINEER Drawing

1. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
2. Select **Medusa (s.*, *.she)** in the **Type** box.
3. Select the file you want to import from the list of files.
4. Click **Open**.
5. Scale the appended drawing file to fit the format, move the appended drawing file origin to the lower-left corner, or accept the default location.

Motion Envelope

To Export to Motion Envelope

1. In an open assembly, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **Motion Envlp** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Create Motion Envelope** dialog box and the **Open** dialog box open. In the **Open** dialog box, **Frames file (*.fra)** appears in the **Type** box.
5. Select the Frames file you want to open from the list of available files or browse to find the Frames file. The **Open** dialog box closes.

   Note: Frames files are created in the Mechanical Design application.
6. Complete the **Create Motion Envelope** dialog box and click **Create**.

Neutral Files

Overview of Part and Assembly Neutral Files

About Part and Assembly Neutral Files

A Neutral file is a formatted text file containing information about parts and assemblies created in Pro/ENGINEER. To retain the feature-based design of the part, provide topology, geometry, symbolic parameters and their corresponding values. Pro/ENGINEER formats part geometry data so that other software systems can easily read it and a wide variety of applications can use it. You can create neutral files from parts and assemblies.

Part Neutral files contain information about the part geometry and the features of the part. Assembly Neutral files contain information about how the parts and subassemblies are placed in the assembly and general information about the component parts. When you create a Neutral file from an assembly, the system generates a Neutral file for each component part, and the assembly Neutral file contains a list of the individual part file names.

If the Pro/ENGINEER part or assembly has faceted data, the faceted data is exported to the Neutral file format.

The Neutral file contains information about parts and assemblies which is not supported by IGES (such as attributes). With a Neutral file, you can access this information and use it to create interfaces with other programs.

You cannot open in the earlier versions of Pro/ENGINEER parts and assemblies created in the current version of Pro/ENGINEER. However, you can use the Neutral file format to collaboratively share data with the earlier versions of Pro/ENGINEER. See the Help module Associative Topology Bus for more information.

Assembly Neutral Files

An assembly Neutral file contains a list of the component assembly members and the mass properties of the assembly, as well as the current revision number of the assembly.

Part Neutral Files

Part Neutral files include the feature and geometry data for one part only. Each part Neutral file contains one part entity, one or more feature entities, and as many surface and edge entities as are needed to describe the part geometry. A part Neutral file can also contain other entities such as associated text files and mass properties.

Neutral files follow the naming convention `<objectname>.neu.#`, where `<objectname>` is the name of the part or assembly the system uses to generate the data file and `#` is the version or index of the Neutral file created.

In order to create a Neutral file, you must first retrieve the part or assembly.

Working with Curves Data Format

About Curves Data Format

You can represent each edge by a three-dimensional curve, which is a function of the normalized parameter. The following lists the data formats of the curve types.

Example: Line Data Format

Data format:

end1[3]	(dbl)	Startpoint of the line
end2[3]	(dbl)	Endpoint of the line

Parameterization:

```
(x, y, z) = (1 - t) * end1 + t * end2
```

Example: Arc Data Format

The system defines the arc entity by a plane in which the arc lies. The arc is centered at the origin and is parameterized by the angle of rotation from the first plane unit vector in the direction of the second plane vector. The start and end angle parameters of the arc and the radius are also given. The direction of the arc is counterclockwise if the start angle is less than the end angle; otherwise, it is clockwise.

Data format:

vector1[3]	(dbl)	Defines the plane of the arc
vector2[3]	(dbl)	Defines the plane of the arc

origin[3]	(dbl)	Defines the plane of the arc
`start_angle`	(dbl)	Angular parameter of the startpoint
`end_angle`	(dbl)	Angular parameter of the endpoint
radius	(dbl)	Radius of the arc

Parameterization:

```
t', the un-normalized parameter, is (1 - t) * start_angle + t *
end_angle

(x, y, z) = radius * [cos(t') * vector1 + sin(t') * vector2] + origin
```

Example: Spline Data Format

The spline curve entity is a nonuniform cubic spline, defined by a series of three-dimensional points, tangent vectors at each point, and an array of un-normalized spline parameters at each point.

Data format:

params []	(dbl)	Array of spline parameters (t) at each point
points [][3]	(dbl)	Array of spline interpolant points
tangents[][3]	(dbl)	Array of tangent vectors at each point

Parameterization:

1. x, y, and z are a series of unique cubic functions, one per segment, fully determined by the start, endpoints, and tangents of each segment.
2. Let `p_max` be the parameter of the last spline point. Then, t', the un-normalized parameter, is t * `p_max`.
3. Locate the spline segment such that:

   par_arr[i] < t' < par_arr[i+1]

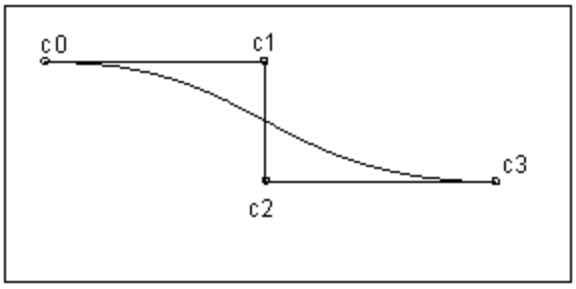
   (if t < 0 or t > +1, use the first or last segment).

   14. t0 = (t' - par_arr[i]) / (par_arr[i+1] - par_arr[i])

   t1 = (par_arr[i+1] - t') / (par_arr[i+1] - par_arr[i])

Example: NURBS Data Format

The system defines the NURBS (nonuniform rational B-spline) curve by expandable arrays of knots, weights, and control points. The following illustration shows a Cubic NURBS Curve.



Data format:

degree	Degree of the basis function
params[]	Array of knots
weights[]	Array of weights for rational NURBS; otherwise, NULL or 1.0 for polynomial b-spline.
`c_pnts`[][3]	Array of control points

Definition:

k = degree of basis function

N = number of knots, degree -2

wi = weights

Ci = control points (x, y, z) * wi

Bi,k = basis functions

By this equation, the number of control points equals N+1.

Working with Trimmed Surfaces Data Format

About Trimmed Surfaces Data Format

The trimmed surface structure contains data that describes the boundary of the surface and the primitive surface that it is on. The primitive surface is a three-dimensional geometric surface parameterized by two variables (u and v). The trimmed surface boundary consists of closed loops (contours) of edges. Edges exist as separate entities in the Neutral file. Each edge is attached to two surfaces, and each edge contains the u and v values of the portion of the boundary that it forms in both surfaces. Surface boundaries are traversed clockwise around the outside of a surface, so an edge has a direction in each surface with respect to the direction of traversal.

You can also find other data in the trimmed surface structure including the rectangular extents of the two-dimensional domain, the three-dimensional surface, and a flag indicating whether the surface normal points towards the inside or outside of the part.

Data format:

id	**(int)**	**Surface ID**
uv_min[2]	(dbl)	Minimum u and v values of the domain
uv_max[2]	(dbl)	Maximum u and v values
xyz_min[3]	(dbl)	Minimum values of a box enclosing the surface
xyz_max[3]	(dbl)	Maximum values of the rectangular box
orient	(int)	1 if the surface normal points outside of the part -1 if it points towards the inside
loops[]	(loop)	Expandable array of loop structures
surface_type	(int)	Constant indicating the primitive surface type
surface	(surface structure)	3D primitive surface data structure

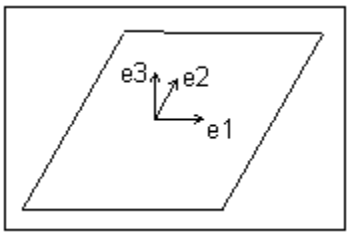
Loop data structure format:

```
edge_ids[ ](int) Array of IDs of the edges in the loop
```

The 3D surface structure of the trimmed surface has a different format for each surface type.

Example: Plane Data Format

The plane entity consists of two perpendicular unit vectors (e1 and e2), the normal to the plane (e3), and the origin of the plane. The following illustration shows a plane entity.



Data format:

e1[3]	(dbl)	Unit vector in u direction
e2[3]	(dbl)	Unit vector in v direction

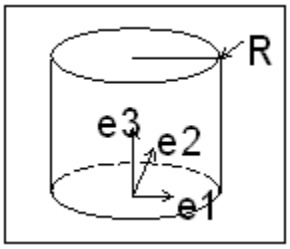
e3[3]	(dbl)	Normal to the plane
origin[3]	(dbl)	Origin of the local coordinate system

Parameterization:

```
(x, y, z) = u * e1 + v * e2 + origin
```

Example: Cylinder Data Format

The generating curve of a cylinder is a line, parallel to the axis, at a distance R from the axis. The radial distance of a point is constant, and the height of the point is v. The following illustration shows a cylinder.



Data format:

e1[3]	(dbl)	
e2[3]	(dbl)	
e3[3]	(dbl)	
origin[3]	(dbl)	Origin of the local coordinate system
R	(dbl)	The radius of the cylinder

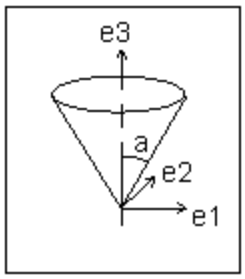
Parameterization:

```
(x, y, z) = R * [cos(u) * e1 + sin(u) * e2] + v * e3 + origin
```

For the cylinder, cone, torus, and the general surface of revolution, the system uses a local coordinate system consisting of three orthogonal unit vectors (e1, e2, and e3) and an origin. The curve lies in the plane of e1 and e3 and rotates in the direction from e1 to e2. The u surface parameter determines the angle of rotation, and the *v* parameter determines the position of the point on the generating curve.

Example: Cone Data Format

The generating curve of a cone is a line at an angle α to the axis of revolution which intersects the axis at the origin. The angle α has a range limitation of . The v parameter is the height of the point along the axis, and the radial distance of the point is `v * tan(`α`)`. The following illustration shows a cone.



Data format:

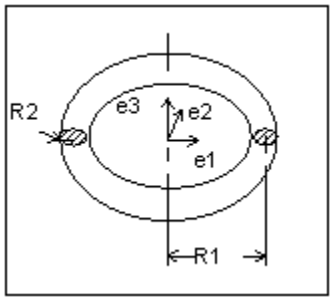
e1[3]	(dbl)	
e2[3]	(dbl)	
e3[3]	(dbl)	
origin[3]	(dbl)	Origin of the local coordinate system
a	(dbl)	The angle between the axis of the cone and the generating line

Parameterization:

```
(x, y, z) = v * tan(α) * [cos(u) * e1 + sin(u) * e2] + v * e3 +
origin
```

Example: Torus Data Format

The generating curve of a torus is an arc of radius R2 with its center at a distance R1 from the origin. R1 cannot equal zero. The starting point of the generating arc is located at a distance R1 + R2 from the origin, in the direction of the first vector of the local coordinate system. The radial distance of a point on the torus is `R1 + R2 * cos(v)`, and the height of the point along the axis of revolution is `R2 * sin(v)`. The following illustration shows a torus.



Data format:

e1[3]	(dbl)	

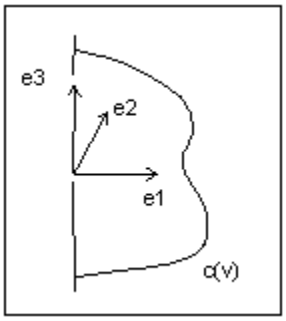
e2[3]	(dbl)	
e3[3]	(dbl)	
origin[3]	(dbl)	Origin of the local coordinate system
R1	(dbl)	The distance from the center of the generating arc to the axis of revolution
R2	(dbl)	The radius of the generating arc

Parameterization:

```
(x, y, z) = (R1 + R2 * cos(v)) * [cos(u) * e1 + sin(u) * e2] + R2 *
sin(v) * e3 + origin
```

Example: General Surface of Revolution Data Format

To create a general surface of revolution, you rotate a curve entity, usually a spline, around an axis. The system evaluates the curve at the normalized parameter v, and rotates the resulting point around the axis through an angle u. The surface of revolution data structure consists of a local coordinate system and a curve structure The curve must be planar; that is, the x coordinate must be greater than or equal to zero and the y coordinate must be zero. The following illustration shows a surface of revolution.



Data format:

e1[3]	(dbl)	
e2[3]	(dbl)	
e3[3]	(dbl)	
origin[3]	(dbl)	Origin of the local coordinate system
C	(curve structure)	Generating curve

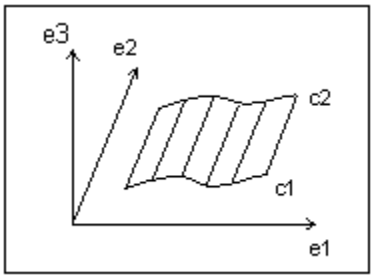
Parameterization:

C (v) = (c1, c2, c3) is a point on the curve.

```
(x, y, z) = [c1 * cos(u) - c2 * sin(u)] * e1 + [c1 * sin(u) + c2 *
cos(u)] * e2 + c3 * e3 + origin
```

Example: Ruled Surface Data Format

A ruled surface is the surface the system generates by interpolating linearly between corresponding points of two curve entities. The u coordinate is the normalized parameter at which it evaluates both curves, and the v coordinate is the linear parameter between the two points. The system does not define the curves in the local coordinate system of the part, so it must transform the resulting point by the local coordinate system of the surface. The following illustration shows a ruled surface.



Data format:

e1[3]	(dbl)	
e2[3]	(dbl)	
e3[3]	(dbl)	
origin[3]	(dbl)	Origin of the local coordinate system
C1	(curve structure)	First generating curve
C2	(curve structure)	Second generating curve

Parameterization:

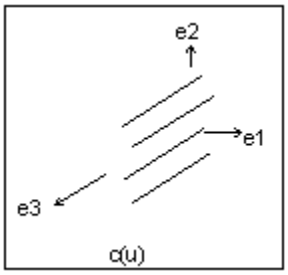
(x', y', z') is the point in local coordinates

```
(x', y', z') = (1 - v) * C1(u) + v * C2(u)
```

```
(x, y, z) = x' * e1 + y' * e2 + z' * e3 + origin
```

Example: Tabulated Cylinder Data Format

The system calculates a tabulated cylinder by projecting a curve linearly through space. It evaluates the curve at the u parameter, and offsets the z-coordinate by the v parameter. It expresses the resulting point in local coordinates and must transform it by the local coordinate system to express it in part coordinates. The following illustration shows a tabulated cylinder.



Data format:

e1[3]	(dbl)	
e2[3]	(dbl)	
e3[3]	(dbl)	
origin[3]	(dbl)	Origin of the local coordinate system
C	(curve structure)	Generating curve

Parameterization:

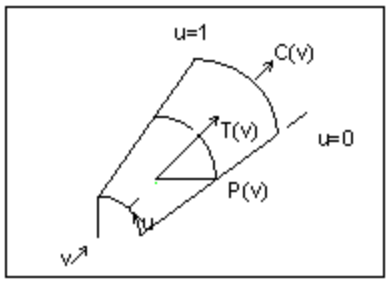
(x', y', z') is the point in local coordinates

```
(x', y', z') = C(u) + (0, 0, v)
```

```
(x, y, z) = x' * e1 + y' * e2 + z' * e3 + origin
```

Example: Fillet Surface Data Format

You can find a fillet surface where a round or a fillet is placed on a curved edge, or on an edge with non-constant arc radii. On a straight edge, you would use a cylinder to represent the fillet. The following illustration shows a fillet surface.



Data format:

`pnt_spline`	P(v) spline running along the u = 0 boundary
`ctr_spline`	C(v) spline along the centers of the fillet arcs
`tan_spline`	T(v) spline of unit tangents to the axis of the fillet arcs

Parameterization:

```
R(v) = P(v) - C(v)

(x,y,z) = C(v) + R(v) * cos(u) + T(v) X R(v) * sin(u)
```

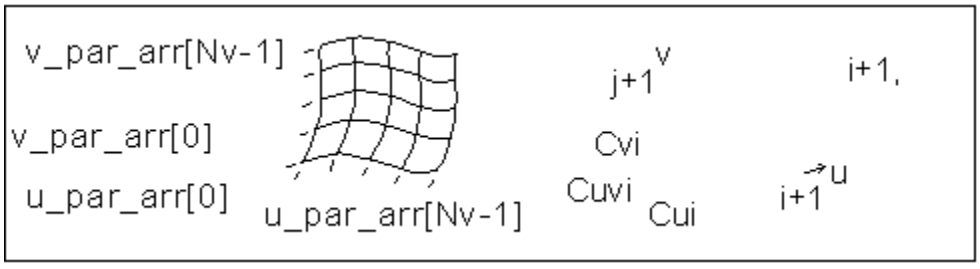
Example: Spline Surface Data Format

The parametric spline surface is a nonuniform bicubic spline surface which passes through a grid with tangent vectors given at each point. The grid is curvilinear in uv space.

The `tan_cond` field, which represents the tangency of the spline surface in the Neutral file format, is an array of two integers. A value of 4 in the `tan_cond` field indicates that the spline surface is periodic in the given direction. Any other integer value indicates that the spline surface is nonperiodic in the given direction.

A cylindrical spline surface includes a spline surface in its definition.

The following illustration shows a spline surface.



Data format:

`u_par_arr`[]	Point parameters, in the u direction, of size Nu

v_par_arr[]	Point parameters, in the v direction, of size Nv
point_arr[][3]	Array of interpolant points, of size Nu & Nv
u_tan_arr[][3]	Array of u tangent vectors at interpolant points, of size Nu & Nv
v_tan_arr[][3]	Array of v tangent vectors at interpolant points, of size Nu & Nv
uvder_arr[][3]	Array of mixed derivatives at interpolant points, of size Nu & Nv

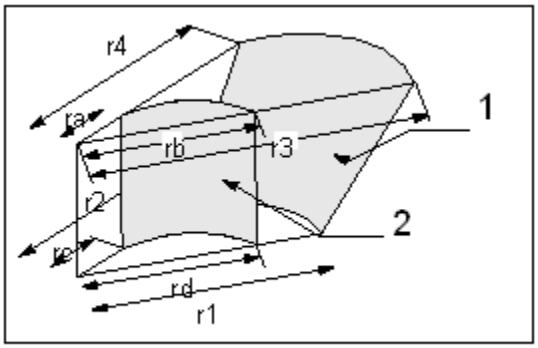
Parameterization:

- Allows for a unique 3 & 3 polynomial around every patch.
- There is second-order continuity across patch boundaries.

Example: Cylindrical Spline Surface Data Format

The cylindrical spline surface is a nonuniform bicubic spline surface that passes through a grid with tangent vectors given at each point. The grid is curvilinear in uv space.

The following illustration shows a cylindrical spline surface.



1 Cone surface S1

2 Cylindrical surface, S0 Spline

Data format:

e1[3]	x' vector of the local coordinate system
e2[3]	y' vector of the local coordinate system
e3[3]	z' vector of the local coordinate system, which corresponds to the axis of revolution of

	the surface
origin[3]	Origin of the local coordinate system
splsrf	Spline surface data structure

The spline surface data structure contains the following fields:

`u_par_arr`[]	Point parameters, in the u direction, of size Nu
`v_par_arr`[]	Point parameters, in the v direction, of size Nv
`point_arr`[][3]	Array of points, in cylindrical coordinates, of size, Nu & Nv. The array components are as follows: `point_arr`[i][0]—Radius `point_arr`[i][1]—Theta `point_arr`[i][2]—Z
`u_tan_arr`[][3]	Array of u tangent vectors in cylindrical coordinates, of size Nu & Nv
`v_tan_arr`[][3]	Array of v tangent vectors in cylindrical coordinates, of size Nu & Nv
`uvder_arr`[][3]	Array of mixed derivatives in cylindrical coordinates, of size Nu & Nv

Engineering Notes

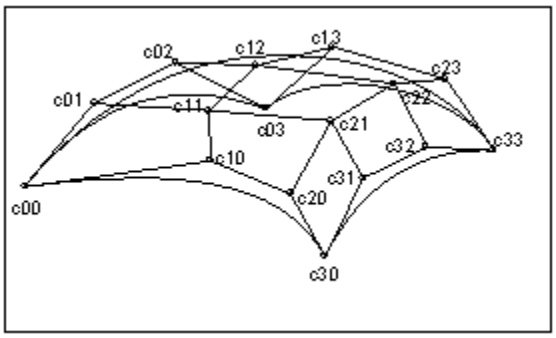
If the surface is represented in cylindrical coordinates (r, theta, z), the local coordinate system values (x', y', z') are interpreted as follows:

- x' = r cos (theta)
- y' = r sin (theta)
- z' = z

You can obtain a cylindrical spline surface, for example, by creating a smooth rotational blend (shown in the figure on the previous page). In some cases, you can replace a cylindrical spline surface with a surface such as a plane, cylinder, or cone. The illustration in Cylindrical Spline Surface on page 5 - 40, shows the cylindrical spline surface *S1* replaced with a cone (r1=r2, r3=r4, and r1r3). If you cannot replace it (such as for the surface S0 in the illustration Cylindrical Spline Surface (ra≠rb or rc≠rd)), leave it as a cylindrical spline surface representation.

Example: NURBS Surface Data Format

Pro/ENGINEER defines the NURBS surface by basis functions (in u and v), expandable arrays of knots, weights, and control points. The following illustration shows a Cubic NURBS surface.



Data format:

deg[2]	Degree of the basis functions (in u and v)
`u_par_arr`[]	Array of knots on the parameter line u
`v_par_arr`[]	Array of knots on the parameter line v
wghts[]	Array of weights for rational NURBS, otherwise NULL
`c_point_arr`[][3]	Array of control points

Definition:

k = degree in u

l = degree in v
N1 = number of knots in u, degree in u -2
N2 = number of knots in v, degree in v -2
Bi,k = basis function in u
Bj,l = basis function in v
wi,j = weights
Ci,j = control points (x,y,z) * wi,j

The weights and `c_points_arr` arrays represent matrices of size wghts[N1+1] [N2+1] and `c_points_arr` [N1+1] [N2+1]. Elements of the matrices are packed into arrays in row major order.

Importing Neutral Files

To Import a Neutral File

1. Click **File** > **Open** (without a part or assembly open) or **Insert** > **Shared Data** > **From File** (with a part or assembly open). The **File Open** dialog box opens.

2. Select **Neutral (*.neu)** in the **Type** box. Neutral files in the working directory appear.
3. Click the Neutral file you want to import from the list of available files or browse to find and then click the Neutral file.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.
5. Accept the default coordinate system or select a coordinate system.
6. Click **Protrusion**, **Cut**, or **Surfaces**.
7. Click **OK** to import the file.

Part Entity Data Format

Each part entity has a list of the surfaces, edges, features, and dimensions associated with it, and the names of any files that contain information related to the part, such as dimension tables, assemblies which include the part, or drawing files.

Data Format:

- surfaces [ ]
- quilts [ ]
- edges [ ]
- datum_curves [ ]
- datum_planes [ ]
- coord_systems [ ]
- attributes [ ]
- layers [ ]
- dimensions [ ]
- features [ ]
- geometric_tolerances [ ]
- colors [ ]
- views [ ]
- xsecs [ ]
- accuracy [ ]

Format of Neutral Files

A Neutral file consists of lines of ASCII text. The format has the following elements:

- A line beginning with the character # is a comment.
- A line not beginning with the character # is of the form:

    - level field value
    - where:
    - level is an integer
    - field is a name
    - value is a string that can be present
- A field must be one of the following:
    - The name of a simple data type (integer, string, real, and so on)
    - The name of an array
    - The name of a structure
    - The name of a pointer to a structure
- All the fields on a particular level belong to the same parent.
- When value is present at a certain level and begins with the left square bracket character `([)`, or consists of a series of digits separated by commas, it indicates that field is an array of dimension `[n]([m] ...)`. The array element values are contained on the lines level+1 up to the next line starting with level again.
    - For one-dimensional arrays, only one line is at the higher level whose value is a string of element values separated by commas.
    - The notation`< n*m>` indicates the following `<n>` elements each have the value `<m>`.
- When value is present at a certain level (and is not the string ->), this is the value to be assigned to field.
- When value is absent at a certain level, this indicates that field is a structure. Immediate elements of the aggregate are contained on lines level+1 up to next line starting with level again.
- The line whose field is the string `ugc_xar_len` is a special case indicating that the next line begins the description of an array of length value.
- When value is the string ->, this indicates that field is a pointer to an aggregate whose elements follow.
- The special value string NULL indicates that the field is a pointer whose value is null.

Edge Entity Data Format

The edge entity contains data about the boundary between two trimmed surfaces. This data includes the IDs of the two surfaces, the direction of the edge with respect to the clockwise boundary traversal of the surfaces, and the uv points of the boundary segment represented by the edge in each surface. The uv points are in groups of four. The first pair of points is the u- and v-coordinates of the edge in

surface 1, and the second pair is the coordinates of the edge in surface 2. Each edge is represented by the same number of points in both surfaces and has a corresponding curve entity included in its data section. The IDs of the two surfaces are consecutive. Individual curves are described below.

Edge Topology Data

Edge topology data consists of all data which describes how the edge represents a specific portion of the boundary between two surfaces.

Data format:

surface_ids[2]	(int)	Identifies surfaces connected to the edge
direction[2]	(int)	Direction of the edge in either surface. This is 1 if the edge is oriented in the same direction as the surface boundary, -1 if it is oriented in the opposite direction.
uv_points[ ][4]	(dbl)	Array of boundary points uv_points[i][0] = u in surface 1 uv_points[i][1] = v in surface 1 uv_points[i][2] = u in surface 2 uv_points[i][3] = v in surface 2
curve_type	(int)	Type of curve represented by the edge
curve	(curve structure)	Corresponding 3D curve
uv_curves [2]	(component structure)	Curves on each side surface

Dimensions Data Format

The dimensions associated with the part are in the following format:

dimensions {Ndim}	At the head of the list
dimensions	For each of Ndim dimensions
name	
value	

tol_plus	
tol_minus	

Feature Entity Data Format

Feature entities appear in sequential order in the Neutral file. A comment line which contains the name of the feature (through hole, round, pattern of slots, and so on) precedes each feature entity data block. The feature header that contains the feature type constant and the ID of the feature appears next. Feature data consists of feature attributes, feature dimensions, and associated entity IDs. The list of dimensions associated with the feature includes the text label of the symbolic dimension and the value of the dimension. A list of the IDs of the surfaces of the feature is also included.

Data format:

features {Nfeats}		At the head of the list
name of feature		Comment line
id	(int)	Feature ID
user_name		String containing wide character types
suppressed	(int)	1 if feature is suppressed 0 otherwise
dim_ids[]	(int)	Array of indices of related dimensions
surface_ids[]	(int)	Array of IDs of feature surfaces
edge_ids[]	(int)	Array of IDs of feature edges
misc_ids[]	(int)	Array of IDs of miscellaneous items; quilts, XYZ-entities

Mass Properties Data Format

The mass properties of an object consist of the revision number for which they are valid; the volume, density, and center of gravity for the object; and the moment of inertia about the x-, y-, and z-axes for the object.

Data format:

revnum	(int)	Object revision number
volume	(dbl)	Volume of the object

density	(dbl)	Average density of the object
c_of_grav[3]	(dbl)	Center of gravity of the object
inertia_x[3]	(dbl)	Moments of inertia about the x-axis
inertia_y[3]	(dbl)	Moments of inertia about the y-axis
inertia_z[3]	(dbl)	Moments of inertia about the z-axis

Geometric Tolerances Data Format

In the Neutral file, Pro/ENGINEER stores the following geometric tolerance information associated with a part: the tolerance class and type, the feature ID tied to the tolerance (and reference IDs if used, if not, the reference ID number is the same as the feature ID), the value of the tolerance, material conditions, and datum reference information (only if used).

Data format:

- geometric_tolerance {Ntols} (At the head of the list)
- geometric_tolerance
- tol_class
- tol_type
- feat_id
- ref_type
- ref_id
- value
- matl_cond
- (datum refs used)

Assembly Neutral Files Data Format

Note: When you create an assembly Neutral file, the system creates files for all part and subassembly subcomponents.

Mass properties of an assembly have the same data format as the mass properties of a part.

Data format:

revnum	(int)	Assembly revision number
massprops		Mass properties of the assembly

member[]		Array of assembly member structures

Assembly Members Data Format

Each assembly member consists of the type of member (either a part or another assembly), the name of the component part or assembly, and the transformation matrix from the component coordinate system to the assembly coordinate system.

Data format:

type	(int)	Either part or assembly
name	(char)	Name of the part or assembly
e1[3]		Transformation matrix from the member coordinate system to the assembly coordinate system
e2[3]		
e3[3]		
origin[3]	(dbl)	

Exporting to a Neutral File

To Export to a Neutral File

1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **Neutral (*neu)** in the **Type** box.
3. Accept the default export filename or type a new filename. It is not necessary to add an extension to the filename. The system creates a file with the name <objectname>.neu.# and stores it in the current working directory.
4. Click **OK**.

Parasolid

About Data Exchange Between Parasolid and Pro/ENGINEER

You can import Parasolid parts and assemblies into Pro/ENGINEER and export Pro/ENGINEER parts and assemblies to the Parasolid format.

Importing from Parasolid

You can directly import 3D geometry such as solids, surfaces, curves, part density, and assembly structure using the Parasolid kernel from a Parasolid-based CAD system in the following native file formats:

- `*.x_t` or `*.xmt_txt`, and `*.xmt` for the text format
- `*.x_b` or `*.xmt_bin` for the binary format
- `*.x_n` or `*.xmt_neu` for the neutral format

Exporting to Parasolid

You can export solids, surfaces, part density, colors, and component structures and names of Pro/ENGINEER parts and assemblies to the Parasolid `*.x_t` text file format.

To Import a Parasolid Part or Assembly

1. Click **File** > **Open** without a file open. The **File Open** dialog box opens.
2. Select **Parasolid (.x_t, .xmt_txt, .x_b)** in the **Type** box. A list of the following Parasolild file formats with their respective file extensions appear in the current working directory:
   - `*.x_t` or `*.xmt_txt` and `*.xmt` for the text format
   - `*.x_b` or `*.xmt_bin` for the binary format
   - `*.x_n` or `*.xmt_neu` for the neutral format
3. Click the name of the Parasolid file in a format that you want to import, or browse to find the file and then click the name of the file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Click **Part** or **Assembly**.
6. Click **OK** in the **Import New Model** dialog box. The imported geometry is automatically positioned in the default location with standard constraints.

To Append a Parasolid Part or Assembly to an Existing Model

1. Click **Insert** > **Shared Data** > **From File** with a file open to append parts as features of an existing part model.

   Click **Insert** > **Component** > **Assemble** to append a Parasolid file as a new part or assembly component of an existing assembly.

   The **File Open** dialog box opens.
2. Select **Parasolid (.x_t, .xmt_txt, .x_b)** in the **Type** box. A list of the following Parasolild file formats with their respective file extensions appear in the current working directory:

- `*.x_t` or `*.xmt_txt` and `*.xmt` for the text format
- `*.x_b` or `*.xmt_bin` for the binary format
- `*.x_n` or `*.xmt_neu` for the neutral format

3. Click the name of the Parasolid file in a format that you want to append to an existing model, or browse to find the file and then click the name of the file.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.

   The **Component Placement** dialog box opens if you are appending a Parasolid file as a new part or assembly component of an existing assembly.
5. Position the appended geometry.

To Export a Part or Assembly to the Parasolid Format

1. Click **File** > **Save a Copy** in a part or assembly. The **Save a Copy** dialog box opens.
2. Select **Parasolid (*.x_t)** in the **Type** box.
3. Browse and select the `*.x_t` file from the working directory or any other location.
4. Accept the default name in the **New Name** box or type a new name for the model.
5. Click **OK**. The **Export PARASOLID** dialog box opens.
6. Specify the structure and contents of the output file in the **Export PARASOLID** dialog box.
7. Click **OK**.

PATRAN

About PATRAN Geometry

PATRAN software systems can read a PATRAN geometry file of a Pro/ENGINEER part. The system formats this file according to the specifications of a PATRAN neutral file. It contains mathematical definitions of items such as surface data which bound a solid. Pro/ENGINEER does not accept files from PATRAN systems as input.

PATRAN files are ASCII text files. The system stores them in the current directory. You can rename or modify them using standard operating system commands. In UNIX, if you create two PATRAN files from the same part, the system overwrites the first one you create unless you rename it.

You can access the **PATRAN Geom** option only in Part mode.

To Export to a PATRAN File

1. In an open part, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.

2. Select **PATRAN (*.ntr)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. In the **New Name** box, accept the default name or type a new model name for the export.
4. Click **OK**. The file exports and the system creates `<filename>.ntr`.

PDF

About Exporting to PDF

You can directly export 2D Pro/ENGINEER drawings, formats and layouts to the Portable Document Format (PDF) from Pro/ENGINEER, using a plug-in. The plug-in requires either an Adobe Distiller license or an Adobe Acrobat license to distill a PostScript (PS) file that is generated by Pro/ENGINEER during the export of Pro/ENGINEER drawings to PDF.

Note: The Adobe Distiller or Adobe Acrobat products and their licenses are not included in the Pro/ENGINEER Wildfire 2.0 installation.

To enable the export of Pro/ENGINEER drawings to PDF, you must ensure the following:

- Adobe Distiller is installed and configured. You can install Adobe Distiller either on the local computer on which you have installed Pro/ENGINEER Wildfire 2.0, or you can access Adobe Distiller from a remote computer.
- The `adobe_distiller` configuration option has been set.
- The required 2D Pro/ENGINEER drawing, format or layout is open and active in the Pro/ENGINEER session.

Distilling PostScript to PDF

Pro/ENGINEER invokes Adobe Distiller in the non-graphics mode and converts the PS file or files to PDF. The temporary PS files created in the default directory for temporary files are deleted after the PDF files are created.

You can generate a PDF document in grayscale or color, depending on the Adobe Distiller settings.

To Export a 2D Object (Drawing, Format, Layout) to PDF

1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **(PDF) *.pdf** in the **Type** box. The existing Pro/ENGINEER 2D object name without the extension appears in the **New Name** box.
3. Accept the default name or type a new name in the **New Name** box.
4. Click **OK**. The Pro/ENGINEER object is exported to PDF.

Note: The PDF file is written to the location you specify. If the specified location is either a Windchill workspace or a remote location, the file is created locally by the distiller, and then copied into the specified location.

5. Alternatively, you can also click  to save the 2D object in the active window to a PDF file. The PDF file is saved in the Pro/ENGINEER working directory.

   or

   Click  to send the 2D object in the active window as a PDF file, through e-mail.

   or

   a. Click **File** > **Print**. The **Print** dialog box opens.

   b. Click  .

   c. Select **Adobe Color PDF** to generate a PDF based on generic color PostScript format, or select **Adobe PDF** to generate a PDF based on generic PostScript format.

   d. Click **OK**.

Pro/DESKTOP

About Importing Pro/DESKTOP Models

The method of directly opening native `.des` part files of Pro/DESKTOP V7 and higher and native `.des` assembly files of Pro/DESKTOP V8 and higher as new models in Pro/ENGINEER is the default. For each feature in the Pro/DESKTOP model, a corresponding read-only feature is created in the directly-opened model in Pro/ENGINEER.

The feature-structure of the Pro/DESKTOP `.des` file is preserved in the resultant Pro/ENGINEER file. The item and feature IDs and configurations in Pro/DESKTOP that correspond to the family table instances in Pro/ENGINEER are preserved in the model. Pro/DESKTOP feature-types that are not supported in Pro/ENGINEER, such as taper-extrusions, do not have the corresponding feature-types in Pro/ENGINEER and are represented by the native Pro/DESKTOP feature icons in the Model Tree.

Note: You must explicitly set the `intf_in_granite_direct_enable` configuration option to `no` to import the `.des` Pro/DESKTOP files to Pro/ENGINEER as import IN features. Unlike the features created by the directly-opened Pro/DESKTOP `.des` models that are read-only, you can modify and regenerate, if necessary, the import IN features and append them to existing Pro/ENGINEER parts.

Regardless of whether `intf_in_granite_direct_enable` is set to `yes` or `no`, you can import only part files of the Pro/DESKTOP `.pdt` file format as import IN features. Opening Pro/DESKTOP 2001 or higher `*.pdt` files as new models in Pro/ENGINEER create import IN features within Conceptual Engineering Data (CED) groups.

Pro/DESKTOP .des and .pdt files create import IN features for the following import and append operations:

- Opening a Pro/DESKTOP model as a new part with **File** > **Open**.
- Assembling a Pro/DESKTOP model as a part component of an existing Pro/ENGINEER assembly with **Insert** > **Component** > **Assemble**.
- Appending a Pro/DESKTOP model as a new part or part component of an existing Pro/ENGINEER assembly using **Insert** > **Shared Data** > **From File**.
- Appending a Pro/DESKTOP model as a new feature of an existing Pro/ENGINEER part using **Insert** > **Shared Data** > **From File**.

If you set the topobus_enable configuration option to yes before directly opening or importing the Pro/DESKTOP files into Pro/ENGINEER,

- The directly-opened .des Pro/DESKTOP models create Translated Image Models (TIMs) that are ATB-enabled at the model level.
- The standard import method creates IN features that are ATB-enabled at the feature level.

Import of Native .des Pro/DESKTOP Files

Native .des Pro/DESKTOP files create a 'group' when you import them as new parts. This 'group' is different from the CED group because this 'group' does not contain the datum planes and datum curves. You can use the Associative Topology Bus (ATB) functionality to update the IN features within this group.

About Conceptual Engineering Data (CED) Groups

Conceptual Engineering Data (CED) groups are created when you import Granite-based .pdt Pro/DESKTOP files to create new models.

Note: Not all .pdt Pro/DESKTOP files create CED groups. Only Pro/DESKTOP 2001 Granite-based .pdt files, created when you export the Pro/DESKTOP file from within Pro/DESKTOP 2001 or higher to the Pro/ENGINEER 2001 format, create CED groups.

Pro/DESKTOP .pdt files create CED groups in the following operations:

- **File** > **Open**
- **Insert** > **Component** > **Assemble**
- **Insert** > **Shared Data** > **From File** into an existing Pro/ENGINEER assembly

A CED group import is different from the standard import of a Pro/DESKTOP .pdt file. A CED group import extracts the sketch plane and profile data from the import feature and constructs native Pro/ENGINEER datum planes and datum curve features from the extracted sketch plane and profile data. The dimensions and constraints used to constrain the profiles in the original Pro/DESKTOP part are used to construct the Pro/ENGINEER curve features. The Pro/DESKTOP IN feature, consisting of only the solid or surface data, and these datum planes and curves, comprise the 'group' called Conceptual Engineering Data (CED).

You can associatively update the entire CED group to changes in the original reference `.pdt` file, but you cannot use the Associative Topology Bus (ATB) Update function to update the CED groups. Instead, you can use **Edit Definition** on the CED group in the Model Tree and the **GROUP HEAD** dialog box that opens to update the CED group with datum planes and datum curves. You need not set the `topobus_enable` configuration option to `yes` to update CED groups.

You can also convert the CED groups or the IN features into standard independent groups or features using **Make group independent** on the **GROUP HEAD** dialog box.

To Open a Pro/DESKTOP File in Pro/ENGINEER

1. Click **File** > **Open** to open a Pro/DESKTOP file as a new model in Pro/ENGINEER. The **File Open** dialog box opens.
2. Select one of the following in the **Type** box:
   - **Pro/Desktop (*.pdt)**
   - **Pro/DESKTOP Direct (*.des)**

   Files of the selected type that are located in your working directory are listed.
3. Browse for the required file if it is not in your working directory.
4. Double-click the name of the part or assembly that you want to import.
5. Click **Open**.
   - By default, Pro/DESKTOP `.des` files directly open as new models, maintaining the original Granite feature history as read-only features because they are Granite-based.
   - If `intf_in_granite_direct_enable` is set to `no`, Pro/DESKTOP `.des` files are opened with standard import functionality and the model geometry or the component model stored in a single import feature.
   - Pro/DESKTOP `.pdt` part files create import IN features even if `intf_in_granite_direct_enable` is set to `yes`.

To Append a Pro/DESKTOP File to a Pro/ENGINEER Model

1. Open a Pro/ENGINEER part or assembly.
2. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
3. Select one of the following in the **Type** box:
   - **Pro/Desktop (*.pdt)**
   - **Pro/DESKTOP Direct (*.des)**

   Files of the selected type that are located in your working directory are listed.
4. Browse for the required file if it is not in your working directory.

5. Double-click the `.pdt` or the `.des` file that you want to add to your model. If one or more coordinate systems exist in the Pro/ENGINEER file, the **Choose Solid Options and Placement** dialog box opens.
6. Accept the default coordinate system in the **Choose Solid Options and Placement** dialog box or use the selection arrow to select another coordinate system.
7. Click to select **Protrusion**, **Cut**, or **Surfaces** as the imported feature when appending a Pro/DESKTOP file to an existing part with solid geometry.
8. Click **OK**. The Pro/DESKTOP file is appended as a new component to a Pro/ENGINEER assembly model or as a protrusion, cut, or quilt to a Pro/ENGINEER part model.

To Assemble a Pro/DESKTOP File Into an Existing Assembly

1. Click **Insert** > **Component** > **Assemble** to assemble a Pro/DESKTOP `.des` model as a part or subassembly component of an existing Pro/ENGINEER assembly. The **File Open** dialog box opens.
2. Select one of the following in the **Type** box:
   - **Pro/Desktop (*.pdt)**
   - **Pro/DESKTOP Direct (*.des)**

   Files of the selected type that are located in your working directory are listed.
3. Browse for the required file if it is not in your working directory.
4. Double-click the file that you want to assemble as a part or subassembly component of an existing Pro/ENGINEER assembly. The **Component Placement** dialog box opens.
5. Position or constrain, or position and constrain, the imported component into the assembly.
6. Click **OK** in the **Component Placement** dialog box.

To Update a Conceptual Engineering Data (CED) Group

1. Select a Conceptual Engineering Data (CED) Group header containing the Pro/DESKTOP import IN feature and the associated curve and datum features from the Model Tree.
2. Right-click. The shortcut menu appears.
3. Click **Edit Definition**. The **GROUP HEAD** dialog box opens.
4. Click **Update geomtools model**.
5. Click **OK**.

ProductView

About Data Exchange Between ProductView and Pro/ENGINEER

You can export a Pro/ENGINEER part, assembly, or drawing to one of the following ProductView file types:

- Viewable `.ol` files that represent parts and part components of an assembly.
- Structure `.ed` files that contain product structure, component position, orientation, and meta data (part- and assembly-level parameter) information.
- Drawing `.plt` plot files that are stored in the HPGL2 format.

  Drawings exported to ProductView create a `.plt` plot file and a `.ed` structure file. The `.ed` file contains the meta data information.

You can import the following data from the `.ol`, `.ed`, and `.edz` ProductView file formats:

- Facet geometry (surfaces and solids)
- Datum geometry (curves, datum planes, datum points, and so on)
- Colors

ProductView `.edz` files are the zipped files of the `.ed` format and are unzipped by the system before import.

Note: ProductView import is not supported on Linux.

To Export a Drawing, Part, or Assembly to a ProductView File Type

1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **ProductView** and one of the following ProductView file types in the **Type** box to export the part, assembly, or drawing to ProductView:
   - Viewable as `.ol`
   - Structures as `.ed`
   - Drawings as `.plt`
3. Accept the default export file name in the **New Name** box or type a new name.
4. Click **OK**.

To Import a ProductView .ol or .ed File into Pro/ENGINEER

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **ProductView (.ol, .ed, .edz)** as the file type in the **Type** box.
3. Select the ProductView file that you want to import from the list of available files or browse to find the file.

4. Click **Open**. A part or an assembly is created depending on the file type you have selected.
   - ProductView `.ol` files create parts.
   - ProductView `.ed` and `.edz` files create assemblies.

To Append a ProductView Part or Assembly to an Existing Model

1. Click **Insert** > **Shared Data** > **From File** with a part or assembly file open. The **File Open** dialog box opens.
2. Select **ProductView (.ol, .ed, .edz)** in the **Type** box. A list of the `.ol` and `.ed` ProductView file formats with their respective file extensions appear in the current working directory.
3. Click the ProductView file or browse to find the file and then click the name of the file.
4. Click **Open**. The **Choose Solid Options and Placement** dialog box opens.
5. Accept the default coordinate system location or select a coordinate system to position the geometry.
6. If you are inserting a feature into a Pro/ENGINEER part that consists of solid geometry, select
   - **Protrusion** or **Cut** to add or subtract the imported geometry from the existing solid.
   - **Surfaces** to insert the imported geometry as a collection of quilts that does not affect the existing solid.
7. Click **OK**.
   - ProductView `.ol` files append as features to the existing part models.
   - ProductView `.ed` files append as part components to existing assemblies.

To Assemble a ProductView Part or Assembly into an Existing Assembly

1. Click **Insert** > **Component** > **Assemble** with an assembly open. The **File Open** dialog box opens.
2. Select **ProductView (.ol, .ed, .edz)** in the **Type** box. A list of the `.ol` and `.ed` ProductView files with their respective file extensions appear in the current working directory.
3. Click a ProductView file or browse to find the file and then click the name of the file.
4. Click **Open**. The **Component Placement** dialog box opens.
5. Add placement constraints to position the part or subassembly component.

6. Click **OK**.
    - ProductView `.ol` files are assembled as new part components.
    - ProductView `.ed` files are assembled as subassembly components

SET

Overview of SET Files

About SET Files

The Standard for Exchange and Transfer (SET) transfers graphics and textual information between computer systems. Pro/ENGINEER supports SET Version 89-06. In SET files, parts and drawings are constructed differently from the way they are constructed in Pro/ENGINEER. There is seldom a one-to-one correspondence between the number and type of entities composing an object in Pro/ENGINEER and the number and type of entities composing the same object in a SET file. This is especially true for parts with blended and swept features.

If you need more information about the use of SET files, see Data Exchange and Transfer Standard Specification (SET), Z 68-300 Version 89-06.

Export/Import of Layers

SET supports the export and import of layers. To export layers to SET from Pro/ENGINEER, you must specify the layer IDs using the **Specify Id** option in the **Setup Layer** menu.

The system exports the layer ID of a drawing entity (geometry, dimension, or note) to a SET file along with the entity. Pro/ENGINEER supports only a single layer per entity through interface files, so if an entity resides in more than one layer, it exports only the highest layer ID (assigned the largest number) as the entity's layer. The system assigns entities with layers read in from SET to a layer in Drawing mode. The layer name of an entity read from SET is `INTF_<nnn>`, where `<nnn>` is the layer ID.

Controlling the Output of Blanked Layers

Set the `intf_out_blanked_entities` configuration option to control the output of blanked layers.

Importing SET Files

To Import a SET File into an Existing Part or Assembly

Note: It is a good practice to create default datum planes before importing SET data.

1. Create or retrieve a model.

    At least one feature must exist in the model.

2. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
3. Select **SET (*.set)** in the **Type** box. SET files in the working directory appear.
4. Click the name of the SET file you want or browse to find and then click the SET file.
5. Click **Open**. The SET Import Log file appears in the **INFORMATION WINDOW**. The **Choose Solid Options and Placement** dialog box opens.
6. Accept the default coordinate system or select a coordinate system.
7. Click **Protrusion**, **Cut**, or **Surfaces**.
8. Click **OK** to import the file.

Importing SET Files

The system creates a log file with a name `< model_name>.log` in your working directory whenever you import a SET model into Pro/ENGINEER. This file contains a summary of SET import processing. You can replace an imported feature with a new import file without having a one-to-one correspondence between existing entities and the replacement entities.

If the SET file contains the necessary topology for surface connection, you can create a solid using **Use Quilt** from the **Solid Opts** menu. If imported files do not have information on surface connection, you can join surfaces in several ways.

Part and Assembly SET Files

When a SET file contains 3D geometry, surface data, and topology, you can import it into a part or assembly to create a nonparametric feature. You cannot modify this feature, but you can add features to the model. If no topology for surfaces exists, even if surface data is present, only a wireframe representation is constructed.

To Import a SET File into a Drawing

1. Create or retrieve a drawing, format, or layout.
2. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
3. Select **SET (*.set)** in the **Type** box. Set files in the working directory appear.
4. Click the filename of a SET file, or browse for the SET file you want.
5. Depending on the size of the imported drawing, the following can happen:
    - If the imported drawing is of a different size than the Pro/ENGINEER drawing, the system prompts you to scale the imported geometry to fit the format. If you type `[N]`, it asks you if you want to move the origin of the imported drawing to the lower-left corner of the format.

      Note: If you did not scale the imported geometry to fit the selected drawing size, you can later replace the drawing format with one of a more appropriate size.

- If the imported drawing is of the same size as the Pro/ENGINEER drawing, the system automatically places the origin of the imported drawing at the lower-left corner (format origin). In this case, no prompt appears.

To Create a Part or Assembly by Importing SET Data

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **SET (*.set)** in the **Type** box. SET files in the working directory appear.
3. Click the name of the SET file you want or browse to find the file, and then click the SET file.
4. Click **Open**. The **Import New Model** dialog box opens.
5. Click **Part** or **Assembly** to indicate the type of model.
6. Accept the default name or assign a new name.
7. Click **OK**. The SET Import Log File appears in the Information Window. The SET data is imported and the new part or assembly is created.

Exporting to SET

To Export a Part or Drawing to a SET File

1. With the part or drawing active, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **SET (*.set)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export SET** dialog box opens.
5. When exporting a part, create or select a coordinate system.
6. Click **OK**.

Exporting Through SET

All SET files created in Pro/ENGINEER follow the naming convention `<objectname>.set`, where `<objectname>` is the name of the currently active part or drawing. If a multisheet drawing is active, the name defaults to `<drawingname>_#.set`, where `#` is the current sheet number. You must create a SET file for each sheet of the drawing.

To Export an Assembly to a SET File

1. With the assembly active, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **SET (*.set)** in the **Type** box.

3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export SET** dialog box opens.
5. Create or select a coordinate system.
6. Specify if you want to output the assembly as one assembly file with internal references or as an assembly file with external references to component files by selecting **Internal ref** or **External ref**.
7. Click **OK**. The assembly exports to a SET file.

   Note: SET does not support all special characters supported by Pro/ENGINEER. Therefore, when you export to the SET file some characters may not appear.

STEP

Overview of STEP Files

About STEP Files

Through STEP (Standard for the Exchange of Product Model Data), you can exchange complete product definitions between heterogeneous computer-aided design, engineering, and manufacturing systems. You can export the combined surface and solid data, and faceted geometry that is in accordance with AP203 and AP214 STEP multiple shape reps and hybrid models. You can export and import:

- Drawings that are in accordance with AP202DIS of the ISO 10303 STEP format.
- Parts and assemblies that are in accordance with AP203IS, AP214CD, AP214IS, and AP214DIS formats.
- Geometry in accordance with the STEP AP209 format.

STEP Format

STEP is an international standard, defined in publications produced by the U.S. Product Data Association (US PRO) IGES/PDES Organization. The complete set of specifications for STEP is referred to as ISO 10303.

The STEP format is organized as a series of documents (referred to as parts in STEP terminology), with each part published separately. Application Protocols (APs), which reference generic parts of ISO 10303, are produced to meet specific data exchange requirements for a particular application. AP202, AP203, and AP214 have reached the status of an International Standard (IS) version. Other APs include AP202DIS, AP209DIS and AP214DIS (Draft International Standard), and AP214 CD II (Committee Draft).

Products supporting STEP can implement this interface using different levels of data transfer. Each level provides various mechanisms to store, accept, and pass product definition data between heterogeneous systems in a consistent and standardized way.

About 3D STEP Validation Properties

When you export to or import from a STEP file, STEP AP203 validation properties are also translated.

Exporting Properties

For solid models exported to STEP, the surface area, volume, and centroid validation properties are translated. Multiple instances of a component in an assembly for centroid are supported. This centroid value is the location of an instance with respect to its parent assembly. For surface models exported to STEP, the surface area validation properties are translated. All assemblies and subassemblies of an assembly are also exported with validation properties.

To include AP203 extensions by default in the file you are exporting to STEP, set the `step_export_format` configuration option to `203_is_ext`. Setting `step_export_format` to `203_is_ext` exports to a STEP file that conforms to the international standard of STEP with the extensions:

- `cla`—Colors and layers
- `gvp`—Geometric Validation
- `ast`—Associative Text

Validation properties are also translated when you set `step_export_format` to `AP214_DIS`.

Importing Properties

If validation properties exist in a STEP file, the validation properties are translated during import. Information about the validation properties is automatically written to the `.log` file. To view the information obtained from the STEP file, open a shell and use a standard text editor to open the `modelname .log` file

Example: .log File for an Imported STEP Component File

This is a portion of a .log file from an imported STEP file. It shows the comparison between data received and computed.

```
Processing information of model AS2_WEDGE_7:
Processing finished at: Thu Jun 3 08:55:30 1999

Total processing time: 00:00:01.
```

	Volume(Inch^3)	**Surface Area(Inch^2**	**CGx**	**CGy**	**CGz for STEP Model**
AS2_WEDGE_7					
As Received	2.060194e+05	3.010194e+04	8.048646e+01	0.000000e+00	1.000000e+01
As Computed	2.060194e+05	3.010194e+04	8.048646e+01	9.457917e-13	1.000000e+01

Difference	2.491288e-08	3.841706e-09	1.574563e-11	9.457917e-13	0.000000e+00
Difference in %	0.000%	0.000%	distance between two centers is		1.577401e-11

About Importing and Exporting STEP Associative Drafting and 3D Data

In Pro/ENGINEER you can import and export STEP AP214 and STEP AP202 Associative drafting data.

During import, Pro/ENGINEER automatically determines if the file is AP214CD2, AP214DIS or AP202IS and translates it appropriately. You do not need to set a configuration option for import.

When you export a STEP file, set the `step_export_format` configuration option to one of the following values:

- `ap203_is` to export 3D models using the `ap203_is STEP` application protocol and conformance class. This is the default in 3D mode.
- `ap214_cd` to export the drawing using the AP214CD2 STEP application protocol and conformance class.
- `ap202_is` to export the drawing using the AP202IS STEP application protocol and conformance class.
- `ap214_dis` (default in drawing mode) to export the drawing using the AP214DIS STEP application protocol and conformance class.
- ap209_dis to export the 3D model using the AP209DIS STEP application protocol and conformance. Use analysis packages for design studies. Does not export edges, boundary conditions, constraints, loads, mesh, and mid planes.
- `ap214_is` to export the 3D model using the AP214IS STEP application protocol and conformance class.

Also set the `step_export_dwg_views` configuration option to one of the following values:

- `no` to export only 2D representations of the 3D models. This is the default.
- `as_3d_views` to export 3D model geometry with its associative views.
- `as_3d_views_assoc_draft` to export 3D model geometry with its associative views and view related annotations.

Supported STEP AP 202 Classes

The interface for STEP provides Level 1 support (file transfer) for the AP202, Associative Drafting including 10 classes. The following table describes the conformance classes that the interface for STEP supports.

Note: The interface for STEP supports all the classes listed during import; however, only classes 1 and 8 are supported for export.

Class	Description
1	Drawing structure and administration plus the presentation of annotation on drawing sheets without views
2	Drawing structure and administration plus the presentation of views depicting the shape, represented by geometrically bounded 2D wireframe models, of one or more products on drawing sheets
3	Drawing structure and administration plus the presentation of views depicting the shape, represented by 2D wireframe models with topology, of one or more products on drawing sheets
4	Drawing structure and administration plus the presentation of views depicting the shape, represented by geometrically bounded 3D wireframe and/or surface models, of one or more products on drawing sheets
5	Drawing structure and administration plus the presentation of views depicting the shape, represented by 3D wireframe models with topology, of one or more products on drawing sheets
8	Drawing structure and administration plus the presentation of views depicting the shape, represented by advanced B-reps models, of one or more products on drawing sheets
9	Drawing structure and administration plus the presentation of views depicting the shape, represented by manifold surface models with topology, of one or more products on drawing sheets
10	Class 2 and the association between dimensions, drafting callouts, and fill area boundaries and their respective target product shape geometry or annotation

Supported STEP AP203 and AP214 Classes

The interface for STEP provides Level 1 support (file exchange) for the AP203, configuration controlled three-dimensional designs of mechanical parts and assemblies. The following table describes the conformance classes that the interface for STEP supports during import or export.

Class	Description
2	Non-Topological Surface and Wireframe as wireframe and surfaces
4	Manifold Surfaces with Topology as shells

6	Advanced Boundary Representation Solids as solids

The interface for STEP supports the following versions of AP214:

- IS (International Standard)
- DIS (Draft International Standard)

Only geometry exchange between Pro/ENGINEER and STEP is supported for this application protocol for both parts and assemblies.

Importing STEP Files to Drawing Mode

To Import a STEP File into Drawing Mode

1. Click **File** > **Open** (without a drawing open) or **Insert** > **Shared Data** > **From File** (with a drawing open). The **File Open** dialog box opens.
2. Select **STEP (.stp, .step)** in the **Type** box.
3. Click the STEP file you want to import on the list of available files or browse to find and then click the STEP file.
4. Click **Open** to create a new model or append the STEP file to the active model. If you are using STEP data to open a new drawing, the **Import New Model** dialog box opens.
5. Click **OK**. The STEP Import Log File appears in the Information Window and the STEP file is imported.

Importing STEP Data in Drawing Mode

When importing STEP data in Drawing mode, Pro/ENGINEER can create an assembly and part only if the imported file contains the data. This assembly is the model of that drawing. If the imported file does not contain an assembly and part, the result is a 2D drawing.

If the drawing sheet size varies from the standard Pro/ENGINEER drawing sheet size, a drawing sheet of a size that is different from the standard Pro/ENGINEER drawing sheet size is automatically created in Pro/ENGINEER.

Note: An AP202 file can contain more than one model.

You can import STEP format files in Drawing mode. (These files are AP202 files and not AP203 or AP214 for part or assembly.) The following entities are supported:

- Sheets
- Sheet annotations (curves, text)
- Views with 3D model projection
- View annotations (curves, dimensions, text)
- Page and view annotations

- 2D hatching
- Standard Pro/ENGINEER symbols as notes
- 2D views
- Multiple-line text as a single multiple-line note

Importing Part and Assembly STEP Files

About Importing Part and Assembly STEP Files

If the model had been exported as a solid from the originating system, Pro/ENGINEER automatically transforms it from quilts to a solid when you import it. When you import STEP geometry and some solid geometry already exists, you can create a solid cut or solid protrusion or you can import them as quilts with the **Choose Solid Options and Placement** dialog box.

When you import a STEP file that contains only surfaces, the system creates quilts out of the surface data upon import. If the STEP file contains surfaces (as opposed to a solid) with all necessary topology for surface connection, you can create a solid using **Make Solid** on the **Properties** menu while redefining the imported feature. If imported files do not have information on surface connection, you can join surfaces in several ways.

If the STEP part or assembly has layers associated with it and you want to make changes to the import status of the layers, set the `intf_in_layer_asm_dialog` configuration option to `yes` to display the **Layer Import Options** dialog box during the import.

To Import Part and Assembly STEP Files

1. Click **File** > **Open** (without a part or assembly open) or **Insert** > **Shared Data** > **From File** (with a part or assembly open). The **File Open** dialog box opens.
2. Select click **STEP (.stp, .step)** in the **Type** box.
3. Select the STEP file you want to import on the list of available files or browse to find and then click the STEP file.
4. Click **Open** to create or append the STEP file.
5. If you are using the STEP data to:
    - Create a part or assembly, the **Import New Model** dialog box opens. Complete the dialog box and click **OK** to open the new file with the STEP data. The STEP Import Log File displays in the Information Window.
    - Append the STEP file to an open part and there are imported features, the **Choose Solid Options and Placement** dialog box opens. Complete the dialog box and click **OK** to accept the imported feature. The STEP Import Log File appears in the Information Window.

6. If the STEP part or assembly has layers associated with it, the **Layer Import Options** dialog box opens.
7. Click **OK** to accept selections made in the dialog box.
8. Click **Close** to close the Information Window.

Importing in 3D

AP203 or AP214 STEP format files can be imported into Pro/ENGINEER in Part or Assembly mode. Although you can import a STEP file that contains Conformance Class 5 entities, Pro/ENGINEER skips these entities during import but imports all other data.

You can import a STEP file from another CAD system that has multiple solid entities in a single part file as an assembly. The interface for STEP imports this file (part file without explicit assembly structure) as an assembly. It imports each solid as a separate part and creates an assembly model into which these parts are placed.

You can replace an imported feature with a new import file without having a one-to-one correspondence between existing entities and the replacement entities.

Exporting Drawings to STEP

About Exporting Drawings to STEP

The STEP AP202 export of 2D drawings enables the transfer of 3D model geometry with its associative views if the `step_export_dwg_views` configuration option is set to `as_3d_views` (default is `no`). It includes 2D entities such as dimensions and other annotations. You can export a 2D drawing and the receiving system can move or add new views.

The STEP AP214 export of 2D drawings enables the transfer of layers, hatches, and special text characters such as degree and diameter symbols as UNICODE instead of separate geometric entities.

To Export a Drawing, Drawing Format, or Layout to STEP

1. With the drawing open, choose **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **STEP (.stp)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The file exports and the system creates `<filename>.stp`.

Exporting Parts or Assemblies to STEP

To Export a Part or Assembly to STEP

1. In a part or assembly, choose **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **STEP (*.stp)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export STEP** dialog box opens.
5. Complete the dialog box to specify the structure and contents of the output file.
6. To export layers, click **Customize Layers** in the **Export STEP** dialog box. The **Choose Layers** dialog box opens.
7. Select **Shells** and **Solids**.
8. Click **OK** to accept the changes made to the **Choose Layers** dialog box.

Exporting Part or Assembly to STEP

Set the `set_export_format` configuration option as follows to export geometry to the STEP AP209 format:

```
set_export_format ap209_dis
```

If you are exporting an assembly, a file named `<name>_asm.stp` and a log file with a name `<name>_out.log.#` are created and stored in the current directory. The `*.stp` file contains a list of the STEP entity types used in creating the exported file.

Configuration Options

The following configuration options support the export of Pro/ENGINEER files to STEP:

- `intf3d_out_extend_surface`
- `intf3d_out_force_surf_normals`
- `intf3d_out_surface_deviation`
- `intf_out_as_bezier`
- `intf_out_assign_names`
- `intf_out_max_bspl_degree`
- `step_export_format`
- `step_appearance_layer_groups`
- `step_in_style_bndry_as_fill_area`
- `export_3d_force_default_naming`

- step_export_ap214_asm_def_mode

File Structure List on Export STEP Menu

When you are exporting an assembly and have set the step_export_format configuration option to apap214_cd, the **File Structure** box appears on the **Export STEP** menu. The option you select determines the file format of Pro/ENGINEER assemblies exported to STEP.

Select File Structure Option	To:	Resulting Default Filename(s)
single_file (default)	Output all assembly geometry to a single file	partname.stp
separate_parts_only	Output all assembly structure to one file, and output parts to individual files	assemblyname_asm.stp partname1.stp, partname2.stp, etc.
separate_all_objects	Output Assemblies, sub-assemblies, and parts to individual files	subassemblyname_asm.stp toplevelassemblyname_top.stp partname1.stp, partname2.stp, etc.

Do not change the default file name(s) because the default file naming conventions differentiate the files created. If you do not want to use the default naming conventions, set the export_3D_force_default_naming configuration option to no (the default). To set the default file format for Pro/ENGINEER assemblies exported to STEP, set the step_export_ap214_asm_def_mode configuration option to one of the file structure options in the table above.

STEP Colors and Layers Support for STEP AP203 and AP214

To export to AP214 STEP format, set the step_export_format configuration option to ap214_cd. The default in 3D mode is ap203_is. If you want a STEP AP203 export to have the same colors and layers as AP214, set the step_appearance_layer_groups configuration option to yes. The default is no.

If the STEP part or assembly has layers associated with it and you want to make changes to the import status of the layers, set the `intf_in_layer_asm_dialog` configuration option to `yes` to display the **Layer Import Options** dialog box during the import.

Pro/ENGINEER Object	Import and Export Type	Layer Supported (Yes/No)	Color Supported (Yes/No)
Solid	Wireframe	Yes	Yes
	Surfaces	Yes	Yes
	Solids	Yes	Yes
	Shells	Yes	Yes
	Datum Curves and Points	Yes	No
Quilts/Surfaces	Wireframe	Yes	Yes
	Surfaces	Yes	Yes
	Solids	Not Applicable	Not Applicable
	Shells	Yes	Yes

The supported export type combinations are:

- Wireframe edges plus datum curves and points. Datum curves and points in the part or assembly are exported in the STEP export file.
- Solids plus shells plus datum curves and points.
- Surfaces plus datum curves and points.

All STEP export types are supported for import. Step import types are not configurable.

Colors

Color support for STEP import and export includes:

- Default colors (colors Pro/ENGINEER assigns to objects when the objects are created)
- System colors (colors the user assigns to objects)
- User-defined colors (colors the user defines and assigns to objects)

You can assign colors to a Pro/ENGINEER part, a surface, all surfaces, quilts and datum curves. You can assign colors to Pro/ENGINEER assemblies, components,

assembly surfaces, assembly quilts and assembly datum curves. There can be a different color assignment for each instance of the same part.

Layers

Layer support for STEP maintains the layer names and their contents represented as the chosen STEP import and export type.

STHENO/PRO

About STHENO/PRO

STHENO/PRO provides a high level of 2D drafting capability for Pro/ENGINEER drawings. You can start STHENO/PRO in the standalone mode or the integrated mode with Pro/ENGINEER. Even without a STHENO/PRO license, you can transfer data between STHENO/PRO and Pro/ENGINEER in the `.tsh` file format through the **File** > **Save a Copy**, **File** > **Open**, and **Insert** > **Shared Data** > **From File** menu operations of export, import, and append.

The integrated mode provides an environment where you can work with multiple Pro/ENGINEER drawings sheets. With a STHENO/PRO license, and a Pro/ENGINEER drawing sheet open, click **Applications** > **Stheno** to start STHENO/PRO in the integrated mode. Pro/ENGINEER and STHENO/PRO run simultaneously and independently on the same host. However, at any given time, only one application is active.

The Pro/ENGINEER drawing sheet is transferred to STHENO/PRO and is read only. However, you can use the suite of drafting tools that STHENO/PRO provides to perform the following sketching and 2D detailing operations:

- Create conceptual sketches
- Merge the Pro/ENGINEER drawing sheet with STHENO/PRO drawings created previously
- Add 2D drawing views
- Select the drawing sheet for reference dimensioning, alignment, and copy

When you exit STHENO/PRO, the additional information is transferred to Pro/ENGINEER as a 2D drawing view. This additional information is placed on the original drawing sheet, reestablishing access to the Pro/ENGINEER drawing and the parametric feature-based model.

To Start STHENO/PRO in Integrated Mode

1. Define the `stheno_root` global environment variable to point to the root directory of STHENO/PRO.
2. Set the `pro_stheno_command` configuration option to point to the STHENO/PRO startup command, `sthenolaunch.exe`.
3. Open a Pro/ENGINEER drawing.

4. Click **Applications** > **Stheno**.

   If no license is available, click **OK** to continue working on the open drawing in Pro/ENGINEER.

   If STHENO/PRO fails to start,

   - Verify that the `stheno_root` global environment variable and `pro_stheno_command` configuration option are correctly set.
   - Exit Pro/ENGINEER and restart it again.

     Note: Exiting Pro/ENGINEER closes STHENO/PRO.

Global Environment Variable for STHENO/PRO

STHENO/PRO application startup command, s`thenolaunch.exe`, finds the location of STHENO/PRO through the `stheno_root` global environment variable.

Define `stheno_root` to point to the root directory of STHENO/PRO in the **Environment Variables** dialog box from **System Properties** in the **Control Panel**.

About Workflow in Integrated Mode

A typical workflow between Pro/ENGINEER and STHENO/PRO follows:

Start Pro/ENGINEER

- Open a Pro/ENGINEER drawing sheet.
- Start STHENO/PRO in integrated mode. The Pro/ENGINEER session suspends and the drawing sheet opens in the STHENO/PRO window.

Work in STHENO/PRO on the Drawing Sheet

- Perform reference dimensioning, aligning, copying, and moving.
- Save the drawing sheet. The system by default saves the drawing sheet with the `.tsh` extension and the drawing sheet number and stores it in the working directory of Pro/ENGINEER.
- Exit STHENO/PRO to resume work in Pro/ENGINEER. You must accept or reject the changes made in STHENO/PRO and choose whether to import the `.tsh` drawing file.

  Note: If you import the `.tsh` drawing file, data in the STHENO/PRO window replaces the STHENO/PRO data in the active Pro/ENGINEER drawing, providing the drawings have the same sheet number.

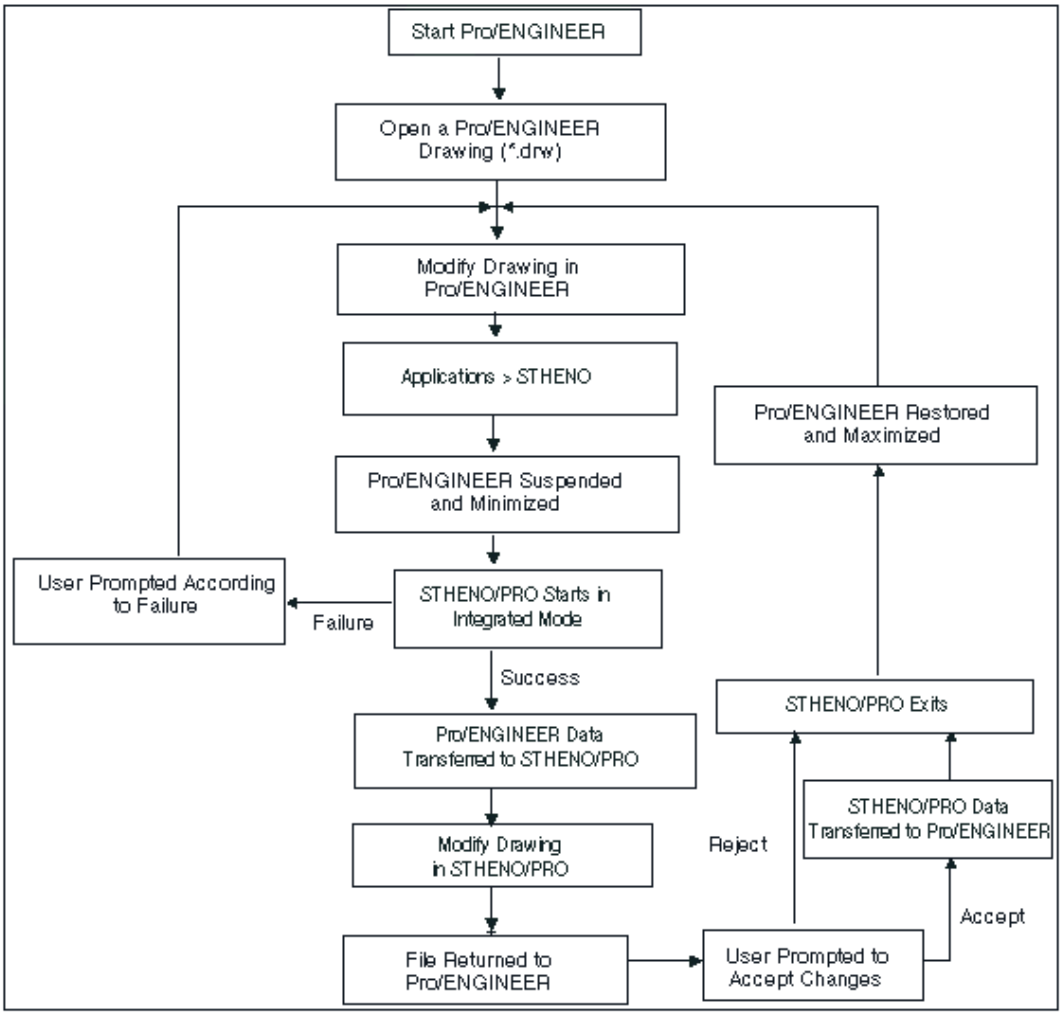
Resume Work in Pro/ENGINEER

- Modify the imported `.tsh` drawing or the `.drw` Pro/ENGINEER drawing file.
- Change the drawing sheets within multiple sheet drawings.

- Transfer the drawing sheet back to STHENO/PRO for further detailing.

Workflow Diagram

The following flowchart shows the typical workflow between Pro/ENGINEER and STHENO/PRO in the integrated mode.



To Import a STHENO/PRO File

1. Click **File** > **Open**. The **File Open** dialog box opens.
2. Select **Stheno (*.tsh)** in the **Type** box. A list of STHENO/PRO files in the working directory appears.
3. Click the name of the STHENO/PRO file that you want to import or browse to find the file.
4. Click **Open**. The STHENO/PRO file opens in a new Pro/ENGINEER drawing sheet.

   You can modify the imported drawing data using the Pro/ENGINEER detailing functionality.

To Append a STHENO/PRO File to an Existing Drawing File

1. With a Pro/ENGINEER drawing file open, click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
2. Select **Stheno (*.tsh)** in the **Type** box. A list of STHENO/PRO files in the working directory appears.
3. Click the name of the STHENO/PRO file that you want to append to the Pro/ENGINEER drawing file or browse to find the file.
4. Click **Open**. The imported `.tsh` drawing file is appended to the Pro/ENGINEER drawing file.

To Export a Pro/ENGINEER Drawing Sheet to STHENO/PRO

1. Open a drawing and click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **Stheno (*.tsh)** in the **Type** box.
3. Accept the default drawing name in the **New Name** box or type a new name for the drawing sheet.
4. Click **OK**.

   You can continue working on the active drawing in Pro/ENGINEER.

Supertab Geometry

About Supertab Geometry

You can use the **SUPRTB Geom** option to create an I-DEAS Supertab file from part data that can be read by Supertab software system. The system formats this file according to the specifications of a Supertab universal file. It contains mathematical definitions of items such as surface data which bound a solid. Pro/ENGINEER does not accept universal files from Supertab systems as input.

Supertab files are ASCII text files. The system stores them in the current directory. You can rename or modify them using standard operating system commands. In UNIX, if you create two Supertab files from the same part, the system overwrites the first one you create unless you rename it.

Note: If you read in the Pro/ENGINEER Supertab file into Geomod, the Geomod software issues a warning for `entity #428`. You must ignore this warning since Geomod does not use the entity.

You can output a Pro/ENGINEER assembly to Supertab in the same way you output parts.

To Export to a Supertab Universal File

1. In an open part, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.

2. Select **SuperTab (*.unv)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The file exports and the system creates `<filename>.unv`.

Tessellated Files

About Tessellated Files

You can export solid model information about parts and assemblies in the following formats: STL (Stereolithography Apparatus), RENDER, Inventor, VRML, OptegraVis, Xpatch, MEDUSA, CatiaFacets (also referred to as Catia Mock-Up), and Pro/CONCEPT.

STL is used for a variety of purposes, the primary one is rapid prototyping.

The RENDER format creates information used to produce high-quality shaded images of Pro/ENGINEER solid models.

Inventor format creates information that can be read by IRIS Inventor, a 3D graphics toolkit by Silicon Graphics.

MEDUSA exports to an ASCII-based `.asc` file. The coordinate systems of the Pro/ENGINEER model that exist on visible layers are exported to MEDUSA with the tessellated geometry.

You can import tessellated files from STL, VRML, DXF, and MEDUSA using **File** > **Open** to create new Pro/ENGINEER models or use **Insert** > **Shared Data** > **From File** to append the files to existing Pro/ENGINEER models.

The tessellated files are imported as faceted data.

Pro/CONCEPT

Pro/ENGINEER outputs the geometry of models for use with Pro/CONCEPT through Wavefront `.obj` files. Wavefront `.obj` files represent geometric information in the form of a triangle of models and quilts. You can output to Pro/CONCEPT in Part and Assembly modes.

File Format

STL and RENDER files represent the surfaces of a solid model as groups of small polygons.

RENDER differs from STL in two respects. RENDER files include:

- The entity color
- A surface normal at each vertex of a polygon element

To Export to an STL, Render, OptegraVis, Xpatch, CatiaFacets, MEDUSA, or Inventor File

1. In an active part or assembly, choose **File** > **Save a Copy**. The **Save a Copy** dialog box.
2. Select **STL**, **Render**, **OptegraVis**, **Xpatch**, **CatiaFacets**, **Medusa** or **Inventor** in the **Type** box. The dialog box specific to file type that you have selected opens.
3. If you want additional control of tessellation, change the values for the maximum **Chord Height** and **Angle Control**.
4. For assemblies only, select:
   - **All Parts** to export the entire assembly.
   - **Include** to select each part you want to include in the file. After you select the parts to include, you can resume selecting by clicking the selection icon next to the message window.
   - **Exclude** to select each part you want to exclude from the file. After you select the parts to exclude, you can resume selecting by clicking the selection icon next to the message window.

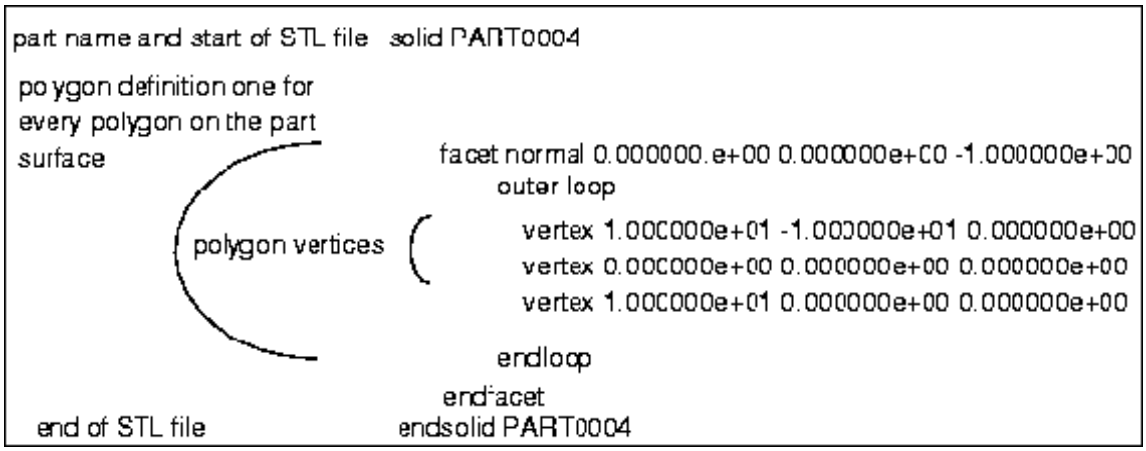
     The message line below these three options indicates the number of parts you are exporting. It is updated each time you exit the **SELECT** menu. Switching between **Include** and **Exclude** does not affect your selections.

     To cancel your selections, click **Reset**.
5. To select or create a coordinate system that defines the xyz space, use **GET COORDS** by clicking the selection icon below **Coordinate System**. You can use the default coordinate system of the model, but this can result in non-positive coordinates. (This option is not available for Inventor.)
6. For STL only, select **Binary** or **ASCII**.
7. Check or uncheck **Allow negative values**. (**Allow negative values** is not available for Inventor, CatiaFacets, MEDUSA or OptegraVis.)
8. Type the name of the file (without the extension) or accept the default name.
9. If problem surfaces are found, fix the edge tessellation problems.
10. Click **Apply** to export the part or assembly or click **OK** to export and close the dialog box.
11. If you are exporting to CATIAFacets, convert the `.cat` file that is created on export to CATIA Native Format.

Example: Samples of Formats for Tessellated Files

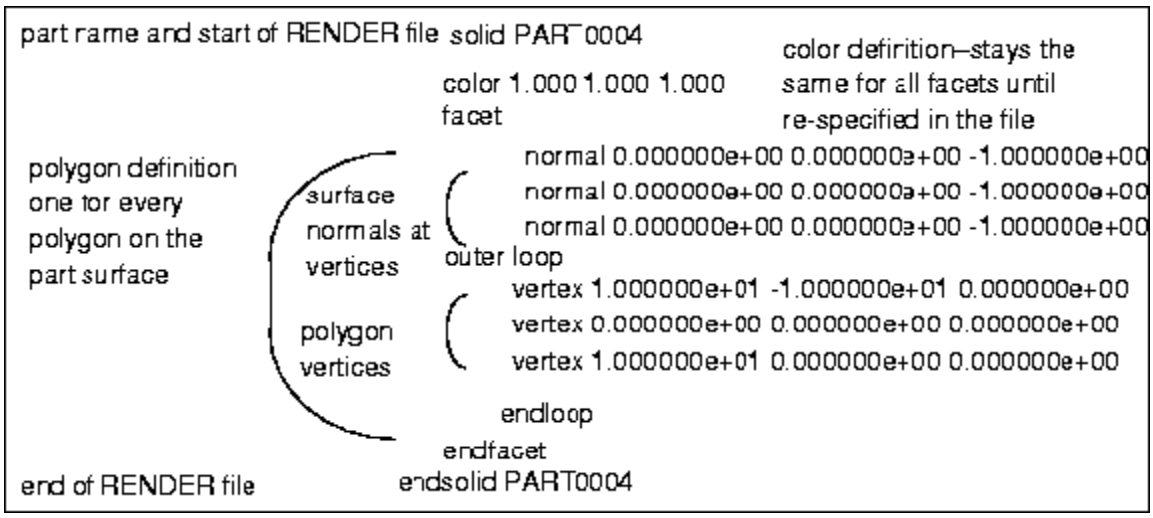
The following illustration shows the file format for an STL file (ASCII).

Format for an STL File



The following illustration shows the format for a RENDER file.

Format for a RENDER File



Creating an Export File

STL, RENDER, and Inventor work only with the parts and assemblies of a Pro/ENGINEER solid model. This means that imported files must be repaired to a solid model first before exporting. In Assembly mode, you can select any number of parts to include in the file.

When you create an STL or Inventor file for a model, the model is shaded and the triangles created are displayed. If the system fails to create an STL file, the problem surfaces are highlighted.

To Export a Model to Pro/CONCEPT

1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **Wavefront (*.obj)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.

3. In the **New Name** box, accept the default name or type a new model name for the export.
4. Click **OK**. The **Export Wavefront** dialog box opens.
5. If you want additional control of tessellation, change the values for the maximum **Chord Height** and **Angle Control**.
6. Click **Add** to select parts and/or quilts to be included in one group.
7. Click **Done Sel**.
8. For the group of objects selected, define the u-v texture mapping. Select a datum plane for planar u-v mapping, or a coordinate system for spherical u-v mapping.

   Note: Selecting a datum plane yields a planar projection for the u-v mapping. Selecting a coordinate system yields a spherical projection onto a sphere, centered at the origin of the coordinate system with the z-axis of the sphere coincident with the z-axis of the coordinate system.
9. Define other groups of objects for output in the same way and specify the corresponding u-v mapping for each group.
10. Click **Apply** to export the part or assembly or click **OK** to export and close the dialog box.
11. Specify the name for the output file (the default name is the model name). A Wavefront `.obj` file containing triangles of all part surfaces and quilts selected for output is generated. The name of the file is `<filename>.obj`. Another file, `<filename>.mtl`, which contains information about the colors you have assigned to the various parts and quilts, is also generated.

Assembly STL and Assembly Inventor File Overlapping Surfaces

Assembly STL and Assembly Inventor files triangulate each part of the assembly individually and place them in the same file.

If you use the STL file to create wax models, overlapping surfaces (where parts are mated) can cause problems. To avoid this, create a single part by merging all of the assembly components, then create an STL file in Part mode.

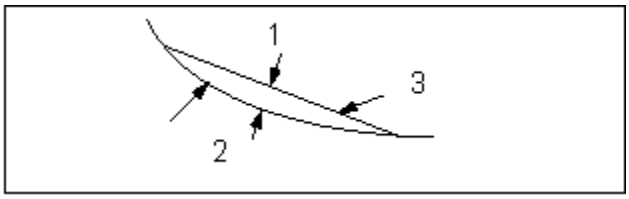
Using Positive Coordinates

Any coordinate system that lies on a part causes some tessellated entities to have non-positive (<= 0) coordinate values. This can cause a problem with programs that only use positive coordinates. To avoid this problem, select a coordinate system that gives all positive values. If the part produces nonpositive values, Pro/ENGINEER notifies you and prompts to continue. If you type `NO`, it quits the process, enabling you to create or select a different coordinate system.

Controlling the Quality of Export

When exporting, you should keep in mind the following:

- To avoid unpredictable results, you should choose a projection that is a one-to-one mapping; that is, no two points of the selected parts or quilts should map to the same point of the datum plane or sphere.
- You can select any part or quilt only once. If you select a quilt or part already included in a group, the system highlights the repeated item and asks you to confirm or cancel the selection.
- The quality of the output file depends on the control of the fineness of triangulation. Control the fineness of triangulation, using:
    - **Chord Height**—Acts as a global specification for tessellating the model surface. It specifies the maximum distance between a chord and a surface. The smaller the chord height specified, the less deviation from actual part surfaces. The chord height, measured in model units, must be within the following range: the lower bound for the chord height is the function of part accuracy, and the upper bound corresponds to the part size. The part size is defined as the diagonal of an imaginary box drawn around the part. The range can be determined by inputting a zero (0) in the message window. This returns the acceptable range for this object in question.



1 Chord height

2 Part surface

3 Tessellated surface

    - **Angle Control**—Regulates how much additional improvement provides along curves with small radii. Specifically, it tessellates curves that have a radius (*r*) defined as:

      `r < r0 = partsize/10`

      to achieve a maximum chord height of:

      `(r/r0)a ChordHeight`

      where α is the **Angle Control** value. Thus, $\alpha = 0$ results in no additional improvement for curves with small radii. When the system bounds a surface feature by a curve with very small radii relative to its part size, such as a dimple on a golf ball, if you do not specify additional improvement using **Angle Control**, these features may have very little definition in the tessellated output. The range is 0.0 to 1.0.

In-Session and Direct VRML Export

About In-Session and Direct Import and Export to VRML

Exporting Pro/ENGINEER parts and assemblies to the VRML format produces a database of VRML files that contain assembly structure, part and assembly names, and geometrical data representing Pro/ENGINEER parts.

It is recommended that you use the in-session export to accommodate maximum model detail.

Pro/ENGINEER allows you to export part and assembly files into VRML 1.0, VRML 2.0, and the ISO Standards file format directly (**File** > **VRML Converter**) or in-session (**File** > **Save a Copy**).

You can import or append VRML 1.0, VRML 2.0, and the ISO Standards file format assembly files using the options **File** > **Open** or **Insert** > **Shared Data** > **From File**. If you are importing formatted assemblies, merge the assemblies before they are converted as part files.

You can view VRML in Pro/FLY-THROUGH and Pro/Model.View products and in other viewers complying with the VRML 1.0, VRML 2.0, and the ISO Standards specifications. To export assembly surfaces, quilts, and welding features to the VRML format, the `shade_surface_feat` configuration option must be set to the default value `yes`.

VRML 2.0 supports "Moving World" through the design animation package.

Considerations when Exporting an Active Part or Assembly

Consider the following if you are exporting an active part or assembly is an in-session export to VRML:

- Regular assembly features are supported only in an in-session export.
- Simplified representations that are currently displayed are supported only in the in-session export. See the `vrml_simprep_export` configuration option for more information.
- The export of cabling data is supported only in the in-session export. The software supports Harness and Cabling components only if you set the environment variable to `Thick Cable`.
- Use the in-session export to export components that have a sublayer in the **DISPLAY** status.
- Export assembly features and exploded states. Only the active exploded state can be exported.
- Export simplified representations with replaced components as a package file.

  Note: Only simplified representations with DEFAULT RULE INCLUDE (master rep) and EXCLUDE RULE for selected components are exported to package files.
- Export a currently displayed `simp. rep`.

Considerations for In-Session and Direct VRML Export

Consider the following information:

- Both in-session and direct VRML export support assembly surfaces and family table instances and retrieval into Pro/ENGINEER.
- You can export simplified representations when the `vrml_simprep_export` configuration option is set to `yes`. Only simplified reps with the DEFAULT RULE INCLUDE (master rep) and EXCLUDE RULE for selected components are exported to package files using either the direct or in-session export.
- When exporting assembly features such as exploded states:
    - You can export a Pro/ENGINEER assembly in the exploded state or in the unexploded state (with exploded features defined).You can use the exploded state for demonstrations and to create images for documentation.
    - If an assembly is exploded at least once during a Pro/ENGINEER session, the exploded data is exported into VRML.

Considerations when Exporting an Inactive Part or Assembly

Exporting an inactive part or assembly is a direct transfer. Consider the following if you are using this method of export to VRML:

- If you make changes to a subassembly, but do not regenerate the large assembly before the direct export, the changes are not reflected in Pro/FLY-THROUGH.
- The direct export is designed for assemblies that are too large or impossible to retrieve (hardware dependent).
- Use Instance accelerator files to improve the export quality when you use the direct transfer export method.

To Prepare the Exploded State for Export

1. In Pro/ENGINEER, retrieve the assembly.
2. Click **View** > **Explode** > **Explode View** to apply the explode state.

   Note: If you want to export the assembly in the unexploded state, after you have applied the explode state, use **Unexplode View**.
3. Export the active assembly.

About the Direct Export Method

The direct export method exports an assembly without retrieving the whole model into memory. Instead, subsets of the model are exported, enabling the export of very large and memory-intensive models. By the end of the direct export process, no parts of the model remain in memory. If changes are made to a subassembly, but the large assembly is not regenerated before the direct export, the changes are not reflected in the model in Pro/FLY-THROUGH. The direct export is designed for large assemblies that are too large or impossible to retrieve (hardware dependent).

Note: If the direct export is used on very large assemblies, you may get draft quality and an inaccurate model representation.

Use the direct export method for an inactive part or assembly.

To Use the Direct Export

1. Set the `direct_vrml` configuration option to `yes`. **VRML Converter** appears on the **File** menu.
2. Click **File** > **VRML Converter**. The **Direct VRML Export** dialog box opens.
3. Click **Browse** (next to **Object Name**) to search for the object that you want to export. The **Source for Direct VRML Conversion** dialog box opens.
4. Select the name of the file you want to convert to VRML format or browse first to find the file and then click the name. The **Source for Direct VRML Conversion** dialog box closes and the **Direct VRML Export** dialog box opens. The file name and path appear in the **Object Name** box.
5. Accept the default **Output Path** or click **Browse** to find and then select another output path. If you browse for an output path, the **Select Working Directory** dialog box opens.
6. Click the output directory for the VRML file.
7. Click **OK**. The **Select Working Directory** dialog box closes and the **Direct VRML Export** dialog box opens. The path name is specified in the **Output Path** box.
8. If you want to change the default settings for the export quality, parameters, or views, click **Setup**. The **VRML Export Setup** dialog box opens.
9. Make changes to the **VRML Export Setup** dialog box.
10. Click **OK** to export the inactive model into VRML format files in the specified folder.

You can use Pro/BATCH to create VRML databases with either direct or in-session export.

VRML Files

For each Pro/ENGINEER part and assembly file, a corresponding VRML file is created. For each part file, a number of VRML level-of-detail (LOD) files are created.

An object is represented graphically by many shaded triangles that make up its form and shape (tessellated format). An LOD is a particular graphical representation that consists of faceted surfaces made up of a certain number of triangles. Multiple levels of files are produced by VRML export, each with a different number of triangles. The more triangles that describe the model, the more detail you see.

The system writes the VRML files to the current Pro/ENGINEER working directory. The software creates the following naming conventions between Pro/ENGINEER files and the assembly and part VRML files:

<assembly_name>.asm exports to <assembly_name>_a.wrl

<part_name>.prt exports to <part_name>_p.wrl

- When simplified representations are exported assemblies that have a non-occurring component, they are named:

  <assembly_name>.asm exports to <assembly_name>_s#.wrl

- When an assembly feature has intersecting components, they are named (part and assembly are modified):

  <assembly_name>.asm exports to <component_name>_af#_a.wrl

  <part_name>.prt exports to <component_name>_af#_p.wrl

  Use the direct export to export components that have a sub layer in DISPLAY status.

The LOD files for each part have the following naming conventions between Pro/ENGINEER files and the LOD VRML files:

<part_name>.prt exports to <part_name>_pr#.wrl

where:

is the number of the particular VRML file. The smaller the number, the greater the LOD in the file.

Note: You can retrieve the <part_name>_pr#.wrl files individually into Pro/FLY-THROUGH.

To Use the In-Session Export

1. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **VRML (*.wrl)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
3. In the **New Name** box, accept the default name or type a new model name for the export.
4. Click **OK**. The **VRML Export Setup** dialog box opens.
5. Make changes to the **VRML Export Setup** dialog box and click **OK** to export the active model into VRML format files in the specified folder.

To Import to VRML Format

1. Click **File** > **Open** without a part or assembly open or **Insert** > **Shared Data** > **From File** with a part or assembly open.

   Click **Insert** > **Component** > **Assemble** to assemble a component into an existing assembly or assemble an assembly as a subassembly component.

   The **File Open** dialog box opens.

2. Select **VRML (*.wrl)** in the **Type** box. The VRML files in the working directory are listed.
3. Click the VRML file you want to import or append from the list of available files or browse to find the required file and then click the `*.wrl` file.
4. Accept the default file name or type a new name in the **Name** box.
5. Click **Open** to import or append the file.

Changing Default Settings for Export to VRML

When you export a direct or in-session file into VRML format from Pro/ENGINEER, the system uses the default setting for the export of the quality (the LOD), parameters, and views. You can change these settings at the time of export clicking **Setup** in the **Export VRML** dialog box.

The **VRML Export Setup** dialog box has the following:

- **Quality**—Enables you to change the LOD export. Choices are `Low`, `Medium`, `High`.
- **Parameters**—Enables you to specify which parameters, if any, you want to export. Choices are `All`, `Designated`, `None`.
- **Views**—Enables you to specify which views you want to export. Choices are `All`, `Top`, `None`.

Configuration File Variables for Export to VRML

Set the following configuration options:

- `shade_surf_feat`
- `vrml_anchor_url`
- `vrml_export_resolution`
- `vrml_parameters`
- `vrml_multiple_views`
- `vrml_simprep_export`

The following table gives examples for the optional keywords.

For a...	To achieve...	Then type vrml_anchor_url...
	WWW anchor node: `http://paran:8000/help`	`http://paran:8000/help`
TOP.asm	WWW anchor node: `TOP_a.wrl`	`$full_name`
TOP.asm	`http://paran:8000/help/TOP_a.html`	`http://paran:8000/help/$name.html`

TOP.asm	http://paran:8000/help/TOP_a.wrl	http://paran:8000/help/$full_name
TOP.asm	http://paran:8000/top/TOP_a.html	http://paran:8000/$base_name/$name.html
TOP.asm	http://paran:8000/TOP.asm	http://paran:8000/$proe_name
part.asm	http://paran:8000/part.asm	

Exporting to VRML Through 4D Navigator

To Export to VRML through 4D Navigator

1. Set the `allow_4dnav_export` configuration option to `yes`.
2. In an active part or assembly, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
3. Select **4D Navigator (*.wrl)** in the **Type** box. The existing Pro/ENGINEER model name without the extension appears in the **New Name** box.
4. In the **New Name** box, accept the default name or type a new model name for the export.
5. Click **OK**. The **Export VRML** dialog box opens.

   If you are exporting an assembly, the **Export VRML** dialog box contains the following additional sections:

   - **Parts to Export**
   - **Format**
6. If you want additional control of tessellation, change the values for the maximum **Chord Height and Angle Control**.
7. For assembles only, click:
   - **All Parts** to export the entire assembly.
   - **Include** to select each part you want to include in the file. After you select the parts to include, you can resume selecting by clicking the selection icon next to the message window.
   - **Exclude** to select each part you want to exclude from the file. After you select the parts to exclude, you can resume selecting by clicking the selection icon next to the message window.

     The message line below these three options indicates the number of parts you are exporting. It updates each time you exit the **SELECT** menu. Switching between **Include** and **Exclude** does not affect your selections.

     To cancel your selections, click **Reset**.

8. For assemblies only, click:
   - **FLAT** to export a single VRML file for the assembly. Geometry in this file is split into multiple parts. For instance parts, each instance is included in the file in its correct position.
   - **ALL PARTS** to export each part in a separate VRML file. Each part is placed in its appropriate position and orientation in the assembly space.
9. To select or create a coordinate system that defines the xyz space, click the selection icon below Coordinate System to access the **GET COORDS** menu. You can use the default coordinate system of the model, but this can result in non-positive coordinates.
10. In the **File name** box, type the name of the file (without the extension) or accept the default name.
11. Click **Apply** to export the part or assembly or click **OK** to export and close the dialog box.

TIFF (Snapshot) Files and Shaded Images

Overview of TIFF Files

About TIFF Files

TIFF supports the interface of digital image data between systems with different architectures. Pro/ENGINEER supports the export and import of TIFF in all graphics modes.

You can use TIFF to store graphical and textual information in a bitmap format and later exchange it between Pro/ENGINEER and different application programs on X Windows on UNIX or GDI on NT. Pro/ENGINEER supports TIFF in Palette color (level 3) mode, making color images available. RGB TIFF can be exported on machines capable of exporting RGB TIFF (machines that run a STARBASE or OPEN GL graphics mode). You can import files in snapshot TIFF format into Pro/ENGINEER only in the Drawing mode.

Importing TIFF Files

To Import a TIFF File

1. Make sure the `graphics` configuration option is set to `x_windows` on UNIX platforms and `win32_gdi` on Windows.
2. Click **Insert** > **Shared Data** > **From File** in an open drawing. The **File Open** dialog box opens.
3. Select **TIFF (*.tif)** in the **Type** box.
4. Select the file name from the list of available files.

5. Click **Open** to import the TIFF file into the drawing.

Considerations when Importing a TIFF File

When you import a TIFF bitmap file into the current drawing:

- The entire image becomes a single entity in Pro/ENGINEER.
- The drawing file is very large when you store it.
- You can manipulate it as you would manipulate normal entities.
- You can print TIFF images imported into a drawing.
- You cannot modify the imported graphics in any way.

Exporting TIFF (Snapshot) Files and Shaded Images

About Exporting a TIFF (Snapshot), TIFF, EPS, or JPEG File

Exporting Shaded Images

You can export wireframe and shaded images in the following formats:

- TIFF
- Shaded Image
- EPS
- JPEG

TIFF and JPEG files are supported by the WWW browser.

You can change the image size, resolution, and depth when you export with TIFF, EPS, or JPEG. When you export with Shaded Image, you export the image based on the current Pro/ENGINEER parameters.

TIFF files are stored in bitmap format and relate to the number of pixels in the image, not the dpi (dots per inch). When you use **Variable** as the size, specify the height and width, and change to a higher resolution, these values do not necessarily represent the actual size of the image.

For example, if you have a 4 inch by 4 inch picture and you output the image at 100 dpi, the image size is 400 by 400 pixels. But if you output the image at 200 dpi, the image size is 800 by 800 pixels. A higher resolution increases the size of the image size.

Use 24-bit RGB to print or export an image with textures. The textures are ignored unless the image uses a 24-bit RGB.

Procedures for Exporting

When exporting to TIFF, Pro/ENGINEER works as a scanner recording the current screen image in bitmaps. An output file represents a snapshot of the entire screen.

You can export files to TIFF (Snapshot) from Part, Assembly, and Drawing modes. TIFF (Snapshot) is the only image to which you can export a drawing file. TIFF files have the extension `.tif`.

To Export to a TIFF (Snapshot) File

1. Click **File** > **Save a Copy** in an open drawing, part, or assembly file. The **Save a Copy** dialog box opens.
2. Select **TIFF (Snapshot) (*.tif)** in the **Type** box.
3. If you want to change the name of the exported image from the current file name, type the new name in the **New Name** box.
4. Click **OK**. The Message `Pop Current Window` appears in the Pro/ENGINEER window.
5. Click **Yes** to export the TIFF file.

To Export a Shaded Image

1. In an open part, or assembly file, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
2. Select **Shaded Image (*.shd)** in the **Type** box.
3. If you want to change the name of the exported image from the current file name, type the new name in the **New Name** box.
4. Click **OK** to export the shaded image. The system adds the extension for the shaded image (`.shd`).

To Export a TIFF, EPS, or JPEG File

1. Click **File** > **Save a Copy** in an open part, or assembly file. The **Save a Copy** dialog box opens.
2. Select **TIFF (*.tif)**, **EPS (*.eps)**, or **JPEG (*.jpg)** in the **Type** box.
3. If you want to change the name of the exported image from the current file name, type the new name in the **New Name** box.
4. Click **OK**. The **Shaded Image Configuration** dialog box opens.
5. Complete the dialog box to specify the dimensions and resolution to use.
6. Click **OK** to export the file. The system adds the `.tif`, `.eps`, or `.jpg` extension.

Shade Image Configuration Dialog Box

Size—Click the **Size** arrow to see a list of standard size options and a **Variable** option and click the size of your choice.

Height—Available only if you click **Variable** from the **Size** List. Type a custom height dimension.

Width—Available only if you click **Variable** from the **Size** list. Type a custom width dimension.

Top Margin—Available only if you click **Variable** from the **Size** List. Type a custom top margin dimension.

Left Margin—Available only if you click **Variable** from the **Size** List. Type a custom left margin dimension

Units—Available only if you click **Variable** from the **Size** List. Click **Millimeters** or **Inches**.

DPI—Determines the resolution output of the image. Click the **DPI** arrow to see a list of DPI values and click the DPI of your choice.

Image Depth—Click the **Image Depth** arrow to see a list of image depth values and click the image depth of your choice.

Unigraphics

About Data Exchange Between Unigraphics and Pro/ENGINEER

You can import Unigraphics `.prt` files as parts and assemblies into Pro/ENGINEER and export Pro/ENGINEER parts and assemblies as Unigraphics `.prt` files. The Unigraphics interface requires a license and an installation of Unigraphics and Pro/ENGINEER on separate workstations or on the same workstation. Install Unigraphics and Pro/ENGINEER on a local workstation or run them from a server machine over the network. However, they must be on the same platform type.

With the Unigraphics interface, you can import and export Unigraphics and Pro/ENGINEER parts, assemblies, part-level layers, and model units.

Unigraphics supports Associative Topology Bus (ATB). See the Help on Associative Topology Bus.

Environment Variables for the Unigraphics Interface

You must set the following environment variables before starting Pro/ENGINEER:

- `UGII_LICENSE_FILE` <`path to the license server for the Unigraphics installation or the node-locked license file`>
- `UGII_BASE_DIR` <`path to the Unigraphics installation directory`>
- `UGII_ROOT_DIR` <`path to the Unigraphics installation directory/bin`> for UNIX and <`path to the Unigraphics installation directory/ugii`> for Windows

The Unigraphics interface requires these environment variables to be set in addition to the UG/Open runtime license and the installation of Unigraphics either on the workstation on which Pro/ENGINEER is installed or accessed over the network.

To Import a Unigraphics Part or Assembly to Pro/ENGINEER

1. Set the `intf3d_ug_install_dir` configuration option to accurately point to the Unigraphics installation.

   Note: If you have not set the `intf3d_ug_install_dir` configuration option properly, the file open operation is aborted.

2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **Unigraphics File (*.prt)** in the **Type** box.
   - By default, the Unigraphics `.prt` files in the working directory are displayed as plain file icons.
   - Pro/ENGINEER `.prt` files are not displayed.

     Note: When you select **Pro/ENGINEER Files (*.prt, *.asm, *.drw, ...)** or **Part (*.prt)** in the **Type** box, Unigraphics files are listed with the Pro/ENGINEER part file icons. When you select a Unigraphics file with one of these types set, a warning in the Pro/ENGINEER window states that the selected file is of Unigraphics origin. It prompts you to select **Unigraphics File (*.prt)** in the **Type** box.
4. Select a Unigraphics `.prt` file from the current directory or browse to select the Unigraphics file from another location.
   - The **Import New Model** dialog box opens.
   - Model type is automatically set to **Part** or **Assembly** based on the contents of the selected file. You can override the default setting of **Part** and change it to **Assembly**, but you cannot change a default setting of **Assembly** to **Part**.
   - For parts, the generic import name is generated for the import model name. The import model cannot be given the same name as the Unigraphics file because Unigraphics and Pro/ENGINEER share the same file extension.
5. Click **OK**.
   - The import log file appears in the Information Window with details of the import.
   - The import model is created.

To Append a Unigraphics File to a Pro/ENGINEER Model

1. Set the `intf3d_ug_install_dir` configuration option to accurately point to the Unigraphics installation.

   Note: If you have not set the `intf3d_ug_install_dir` configuration option properly, the file append operation is aborted.

2. Open a Pro/ENGINEER part or assembly.

3. Click **Insert** > **Shared Data** > **From File**. The **File Open** dialog box opens.
4. Select **Unigraphics File (*.prt)** in the **Type** box.
   - By default, the Unigraphics `.prt` files in the working directory are displayed as plain file icons.
   - Pro/ENGINEER `.prt` files are not displayed.

     Note: Unigraphics `.prt` files are not displayed if you select **All Files (*)** in the **Type** box.
5. Select a Unigraphics `.prt` file from the current directory or browse to select the Unigraphics file from another location.
   - The **Choose Solid Options and Placement** dialog box opens if you are appending the Unigraphics file as a part.
   - Accept the default coordinate system in the **Choose Solid Options and Placement** dialog box or use the selection arrow to select another coordinate system to locate the Unigraphics data.
   - If the Pro/ENGINEER part file already contains solid geometry, specify **Protrusion**, **Cut**, or **Surfaces** in the **Choose Solid Options and Placement** dialog box to create a protrusion or cut from the imported geometry or represent the geometry as a collection of quilts.
   - Click **OK** in the **Choose Solid Options and Placement** dialog box. The Unigraphics `*.prt` file is appended as a protrusion, cut, or quilt to the Pro/ENGINEER part.
6. If you are appending the Unigraphics file as a part component to an existing Pro/ENGINEER assembly containing at least one coordinate system, select a coordinate system in the **SEL COORD S** menu. The Unigraphics `*.prt` file is appended as a new component to the Pro/ENGINEER assembly.

To Assemble a Unigraphics File into a Pro/ENGINEER Assembly Model

1. Set the `intf3d_ug_install_dir` configuration option to accurately point to the Unigraphics installation.

   Note: If you have not set the `intf3d_ug_install_dir` configuration option properly, the file assemble operation is aborted.
2. Open a Pro/ENGINEER assembly.
3. Click **Insert** > **Component** > **Assemble**. The **File Open** dialog box opens.
4. Select **Unigraphics File (*.prt)** in the **Type** box.
   - By default, the Unigraphics `.prt` files in the working directory are displayed as plain file icons.
   - Pro/ENGINEER `.prt` files are not displayed.

Note: When you select **Pro/ENGINEER Files (*.prt, *.asm, *.drw, ...)** or **Part (*.prt)** in the **Type** box, Unigraphics files are listed with the Pro/ENGINEER part file icons. When you select a Unigraphics file with one of these types set, a warning states that the selected file is of Unigraphics origin. It prompts you to select **Unigraphics File (*.prt)** in the **Type** box. Unigraphics `.prt` files are not displayed if you select **All Files (*)** in the **Type** box.

5. Select a Unigraphics `.prt` file from the current directory or browse to select the Unigraphics file from another location.
6. The **Component Placement** dialog box opens.
7. Accept the default coordinate system location or select a coordinate system to position the geometry.
8. Package the imported part or subassembly component into the assembly or add associative placement constraints.
9. Click **OK** in the **Component Placement** dialog box.

To Export a Pro/ENGINEER Part or Assembly to Unigraphics

1. Set the `intf3d_ug_install_dir` configuration option to accurately point to the Unigraphics installation.
2. With part or assembly open, click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
3. Select **Unigraphics File (*.prt)** in the **Type** box. The **Save a Copy** dialog box displays the existing Unigraphics file in the working directory.
4. Browse to select a location for the file after export. Existing Unigraphics `.prt` files in the directory are displayed.
5. Accept the default name with the extension, `_ug_prt` or `_ug_asm` that is automatically added to the exported part or assembly name, respectively, in the **New Name** box or type a new name for the exported model.
6. Click **OK** in the **Save a Copy** dialog box.

   Note: If you have not set the `intf3d_ug_install_dir` configuration option to point to the Unigraphics installation directory, a warning in the Pro/ENGINEER window prompts you to specify the location of the Unigraphics installation.

   - The Pro/ENGINEER file is exported to the location you have specified.
   - For an assembly, the exported component model files are named after the corresponding Pro/ENGINEER component model name with the extensions, `_ug_prt` and `_ug_asm`, depending on the model type.

VDA

Importing VDA Files

To Create a Part by Importing VDA Data

1. Click **File** > **Open** when creating a part. The **File Open** dialog box opens.
2. Select **VDA (*.vda)** in the **Type** box. VDA files in the working directory appear.
3. Select the VDA file you want to import from the list of available files or browse to find the VDA file.
4. Click **Open**.
5. The **Import New Model** dialog box opens. Accept the default file name or type a new name.
6. Click **OK**. A `.log` file appears in the INFORMATION WINDOW. The file is processed. The part is created automatically, placing at the VDA data origin a coordinate system named after the VDA file name.

To Import a VDA File as a New Feature into an Existing Part

1. Create or retrieve a part (at least one feature has to exist).
2. Click **Insert** > **Shared Data** > **From File.** The **File Open** dialog box opens.
3. Select **VDA (*.vda)** in the **Type** box.
4. Click the VDA file you want to import from the list of available files or browse to find the VDA file.
5. Click **Open**.
6. The Information Window and the **Choose Solid Options and Placement** dialog box open.
7. In the **Choose Solid Options and Placement** dialog box, create or select a coordinate system to locate the VDA data.
8. Click **OK**. The feature is created.

Importing Through VDA

A log file is created in your working directory with the name `<model_name>.log` when you import a VDA model into Pro/ENGINEER. This file contains a summary of the VDA import process.

Pro/ENGINEER can read a VDA file into an existing part, or you can use the VDA data to create a new part. The type of features that the VDA file creates depends on the data available. It creates solid features only if you provide a topologically complete description using the VDA entities TOP, FACE, CONS, and CURVE.

You can set the `read_vda_in_pset_as_spline` configuration option to control the data point sets imported from a VDA file.

You can replace an imported feature with a new import file without having a one-to-one correspondence between existing entities and the replacement entities.

If the VDA file contains information for surface connection, you can create a solid using **Join Surfs** on the **Properties** menu. If imported files do not have information on surface connection, you can join surfaces in several ways.

Exporting to VDA

To Export a Part or Assembly to a VDA File

1. Click **File** > **Save a Copy** with the assembly active. The **Save a Copy** dialog box opens.
2. Select **VDA (*.vda)** in the **Type** box.
3. Accept the default name in the **New Name** box or type a new model name for the export.
4. Click **OK**. The **Export VDA** dialog box opens.
5. Under **Export**, indicate the type of entities to export by selecting the following:
   - **Wireframe**—Outputs part edges only.
   - **Surfaces**—Outputs all part surfaces and surface boundaries.
   - **Datum Curves and Points**—Outputs datum entities.
6. Specify the structure and contents of the output file.
7. If you want to export layers, click **Customize Layers** in the **Export VDA** dialog box. The **Choose Layers** dialog box opens.
8. Click **Auto ID** to assign layer IDs to layers that do not have IDs.
9. Make changes to the **Choose Layers** dialog box and click **OK** to accept the changes made.

   Pro/ENGINEER creates the VDA data file. The origin of the data defaults to the assembly origin. When the file has been created, you are notified.

Exporting to VDA

Before creating a VDA file, you may want to establish a standard VDA file header.

The system enters the VDA version, sender's name, and date automatically. The sender's name is the user's login ID. To fill in the other information, create a header file and specify its name using the `vda_header <filename>` configuration option complete with the extension, if any.

All VDA files created in Pro/ENGINEER follow the naming convention `<objectname>.vda`, where `<objectname>` is the name of the currently active

assembly. In UNIX, if you create two VDA files from the same object, the first VDA file is overwritten by the file you create unless you rename it.

Using Pro/BATCH Utilities

About Pro/BATCH

The Pro/BATCH user interface enables you to define your batch session and schedule the execution of the command file.

You can upgrade files from a previous release of Pro/ENGINEER to the current version of Pro/ENGINEER.

Pro/BATCH enables you to enter batch mode to create multiple plot files, or export to other formats. Using the interface, or a command file that you create, Pro/ENGINEER retrieves the appropriate objects and creates the specified export files for them. You can also import CADAM data into Pro/ENGINEER using Pro/BATCH.

The `proigsutil` conversion utility enables you to convert a number of IGES files into parts or drawings.

Batch mode takes a list of objects from which to create files, retrieves the objects, and creates the files without any further interaction.

When running Pro/BATCH with the interface or the command line,

- License requirements and available modules remain the same as for a regular Pro/ENGINEER session.
- Pro/BATCH does not remove a version number from an input file. If you supply a version number, the system uses the file with that version number; if no version number is supplied, the latest version of a file is used.
- The configuration file in the working directory, home directory, loadpoint directory, or startup directory is used when executing a command file. The configuration file settings define the Pro/ENGINEER environment and specify options that are required to perform a specific task (for example, plotting or exporting to IGES, DXF, or VDA).
- The configuration settings remain the same for all files created from the same batch job. Because the system also uses search paths defined in the configuration file, objects need not reside in the current directory.

When running Pro/BATCH with the command line,

- If you press ENTER, the system accepts the default value given within the brackets ([ ]).
- To exit Pro/BATCH while creating a command line, type QUIT + ENTER + ENTER.

A log file is created for each Pro/BATCH session. The log contains all of the executed commands as well as the exit code for the command (SUCCESS or FAILURE).

About Distributed Pro/BATCH

Pro/ENGINEER includes another batch processing capability called Distributed Pro/BATCH. You can use Distributed Pro/BATCH to perform time-consuming and routine tasks, such as,

- Plotting and printing
- Importing and exporting 2D and 3D data from supported interfaces
- Creating shaded images of Pro/ENGINEER parts and assemblies
- Checking models using ModelCHECK
- Saving Pro/ENGINEER models from the earlier versions in the latest version

You can perform batch operations not only on a local computer (standalone mode), but also on multiple computers on the network (distributed mode). In the distributed mode, Distributed Pro/BATCH used the PTC Distributed Services Manager (DSM) to administer and manage remote computers on the network. DSM is a separate network-based desktop application.

You can install Distributed Pro/BATCH from the Pro/ENGINEER CD-ROM and is available under **Options** for Pro/ENGINEER. For more information on Distributed Pro/BATCH, see the Help module Distributed Pro/BATCH.

Example: .pro_batch.log#file

Recording the results of three operations (plotting, exporting to IGES, and exporting to VDA).

```
-plot -object shaft.prt -plotter default - paper A
          -plotfile plot.plt

          SHAFT - success

          -----   Result SUCCESS

          -iges -object shaft.prt

          -----   Result SUCCESS

          -vda -object shaft.prt

          -----   Result SUCCESS
```

To Enter Pro/BATCH and Set Preferences (Basic Steps)

1. Type `pro_batch`. The **Pro/BATCH** dialog box opens.
2. Click **Preference** > **Set Preference**. The **Option Preference** dialog box opens.

3. Click the **General** tab. Set the default action for all the objects that you add to the **Pro/BATCH** dialog box
4. In the **Default Action** box, select a default action.

   The **Option Preference** dialog box settings are saved in your home directory when you close the main window.

   Note: If you include objects for plotting in the batch file, you must enter the Plotter Command on the Plot page.
5. Click other tabs for which you want to enter information.
6. Click **OK** to close the dialog box.

To Schedule the Start of the Batch Process

1. Before exiting the **Pro/BATCH** dialog box, click **Schedule** > **Start the task**. The **Schedule** dialog box opens.
2. In the hour(s) box, click the arrows to select the number of hours.
3. Click **OK**.

   Note: To run the batch file immediately, use the default of zero hours.

   At the scheduled time, Pro/BATCH starts processing and creates the following files in your current working directory:

   - Object output files (for example, `<object>.plt`, `<object>.iges`)
   - The `pro_batch_list.txt` file listing all object output files
   - The `pro_batch.log.#` file recording the results of the execution.

To Upgrade a File to the Version of the Current Release

1. On the menu, click **upgrade**.
2. Schedule the start of the batch process.

To Create a New BATCH File

1. Type `pro_batch`. The **Pro/BATCH** dialog box opens.
2. In the **Batch File** box, type the name of the new batch file.
3. In the **Pro/Engineer Command** box, type the Pro/ENGINEER command to run the batch file.
4. Click **File** > **Browse**. The **File Browser** dialog box opens.
5. In the **Filter** list, select a file type. A list of all the files of the type selected appears.

6. From the list of files, select the name of the file you want to include in the batch file and click **Add Objects**. The file appears in the **Pro/BATCH** dialog box. Repeat steps 5and 6 for all the files that you want to add to the batch file.
7. Click **Close Browser**.
8. Select the name of the file that you want to set up for a particular process in your **Pro/BATCH** dialog box. Do either of the following:
   - Click **Options** > **Set Action**.
   - Right-click. A shortcut menu appears. Click **Set Action**.

   A menu appears with the following:
   - **plot**
   - **stl**
   - **iges**
   - **vda**
   - **step**
   - **dxf**
   - **cgm**
   - **clfile**
   - **vrml**
   - **upgrade**

To Complete the Set Option - cgm Dialog Box

1. From the menu, click **cgm**.
2. Click **Options** > **Set Option**. The **Set Option - cgm** dialog box opens.
3. Click **Cleartext** or **Milspec** for the export type of the `cgm` file.
4. Click **Abstract** or **Metric** for the type of scale value.
5. In the **Output Filename** box, type the output file name for the `cgm` file.
6. Click **OK** to define the setting for the `cgm` file. The option changes from default to user-defined in the **Pro/BATCH** dialog box.
7. Click **File** > **Save** or **File** > **Save As**.
8. Schedule the start of the batch process.

To Complete the Set Option - plot Dialog Box

1. From the menu, click **plot**.

2. Click **Options** > **Set Option**. The **Set Option - plot** dialog box opens.
3. In the **Plotter** list, select the plotter of your choice.
4. In the **Paper Size** box, select the necessary paper size for the plot.
5. In the **Output Quality** box, select a value. The value you select for quality supersedes the setting for `interface_quality` in your configuration file.
6. In the **User Scale** box, type a value for the scale you want.
7. Click **Pen Slew** or **No Pen Slew**. If you select **Pen Slew**, specify the pen speed.
8. Click **Segment** or **Labeled**.
9. If your drawing has multiple sheets, select the page range to plot.
10. Click the type of output file and type the name of the plot file you are creating in the **Filename** box.
11. Click **OK** to define the setting for the plot file. The Option changes from default to user-defined in the Pro/BATCH dialog box.
12. Choose **File** > **Save** or **File** > **Save a Copy**.
13. Schedule the start of the batch process.

    Note: The printer command in the **Option Preference** dialog box must be set.

To Complete the Set Option - stl Dialog Box

1. From the menu, click **stl**.
2. Click **Options** > **Set Option**. The **Set Option - stl** dialog box opens.
3. Click **binary** or **ascii** for the output of the `stl` file.
4. In the **Output Quality** list, select a value for the output quality of the file.
5. In the **Filename** box, type a file name for the `stl` file.
6. Click **OK** to define the setting for the type of `stl` file output. The **Option** changes from default to user-defined in the **Pro/BATCH** dialog box.
7. Click **File** > **Save** or **File** > **Save a Copy**.
8. Schedule the start of the batch process.

To Complete the Set Option - vrml Dialog Box

1. From the menu, click **vrml**.
2. Click **Options** > **Set Option**. The **Set Option - vrml** dialog box opens.
3. Click **In Session Export** or **Direct Export**.
4. In the **Output Path** box, type the path for the `vrml` file. For example:

`-vrml -object <object name> -out_path <path> [-direct y]`

where:

`<object name>`—A part or assembly, such as `wheel_1.prt` or `engine_u.asm.2`.

`<path>`—Path to optionally divert `wrl` files to specified location.

`[-direct y]`—A direct export.

5. Click **OK** to define the setting for the `vrml` file. The option changes from default to user-defined in the **Pro/BATCH** dialog box.
6. Click **File** > **Save** or **File** > **Save a Copy**.
7. Schedule the start of the batch process.

To Complete the Set Option Dialog Box for IGES, DXF, SET, VDA, or STEP

1. Click **iges**, **set**, **vda**, **step**, or **dxf**.
2. Click **Options** > **Set Option**. The **Set Option** dialog box opens for the selected file type.

   Note: If you are creating a DXF file, you can only export a drawing.
3. Click **OK** to define the setting for the specified file type. The option changes from default to user-defined in the **Pro/BATCH** dialog box.
4. Click **File** > **Save** or **File** > **Save a Copy**.
5. Schedule the start of the batch process.

Configuration File Options Used by Pro/BATCH

Pro/BATCH recognizes the following configuration options:

- `search_path`
- `pro_plot_config_dir`—The `.pcf` file not in the working directory.
- `plot_file_dir`—The output directory into which the plot files are placed instead of the current directory by default.
- `plotter_command`—The command defined to plot the output files.

Using a Plotter Configuration File When Plotting with Pro/BATCH

If you want to use a plotter configuration file to specify plotting options, type the name of the file (that is, `<filename>.pcf`) when prompted to specify an output device. The system uses the plotting options defined in the `.pcf` file and does not prompt you for them. The following plotter configuration file options are valid in Pro/BATCH:

- `interface_quality`

- paper_size
- pen_slew
- plotter
- plot_label
- plot_scale
- plot_segmented
- plot_sheets

Outputting to Other Formats Through Pro/BATCH

About Outputting to Other Formats Through Pro/BATCH

The following table summarizes Pro/BATCH capabilities for exporting Pro/ENGINEER data to other formats and lists the configuration file options controlling the contents of an output file.

Output Format	Supported in Part	Supported in Assembly	Supported in Drawing	Configuration File Options Controlling Export
IGES	•	•	•	assemblies– iges_out_assembly_default_mode 3D data– intf3d_out_default_option and intf3d_out_datums_by_default
DXF			•	
STEP	•	•		3D data– intf3d_out_default_option and intf3d_out_datums_by_default
VDA	•			
SET	•	•	•	
STL	•	•		
CGM	•	•	•	

The following configuration options control the output functions and file contents:

- intf3d_out_default_option
- intf3d_out_datums_by_default
- iges_out_assembly_default_mode

Importing from CADAM with Pro/BATCH

About Importing from CADAM with Pro/BATCH

You can import CADAM drawings with Pro/BATCH if you specify cadamin when creating an executable. You are prompted to specify the following:

- Group name
- Subgroup name
- CADAM drawing name
- Pro/ENGINEER drawing name (default is the CADAM drawing name)

The name for the Pro/ENGINEER drawing cannot have spaces. If spaces are detected, you are prompted to specify a new name.

After you have specified all necessary data, the system adds a -cadamin to the batch file. The general form of the command is as follows:

```
-cadamin -grp_nam <group> -sgrp_nam <subgroup> -dwg_nam <name> -
pro_dwg_name
```

When modifying the executable, you must type the arguments "-grp_nam", "-sgrp_nam", and "-dwg_nam" with quotes.

Using a Pro/BATCH Command File

About Pro/BATCH Command Files

The command file is a list of objects that you use to create a plot file or to export to other formats. Each line in the command file includes the name of an object to be exported.

Executing a Command File

Executing a command file creates the files for the objects specified in the command file. It also creates the pro_batch_list.txt file containing a list of all the files successfully created. You can use the resulting file within a script to plot all of the plot files automatically with a single command. You can control hidden lines and datum display in batch mode through your configuration file.

To Create a New Command File

1. Type pro_batch -text <pro_command>. A message appears asking if you want to run an existing command file

2. Type `no`.
3. In the **Command File** box, type a command file name. If you type the name of an existing command file, a message appears asking if you want to overwrite the existing file. Type `no` if you want to use a different command file name.
4. Type the name of the type of command line you are creating (for example, plot, STL, DXF, or IGES).
5. Depending on your task, use the appropriate procedure.

To Execute a Command File Immediately After It Is Created

1. Create new command files as required.
2. When the message appears asking if you want to create another command line type `no`.
3. Type `YES` to execute the batch command file. A message appears asking if you want to plot the output files.
4. Do one of the following
   - Press ENTER to accept `no`, the default
   - Type a new system command (for example, `laser`)
   - Type the default `plotter_command` if the `plotter_command` is specified in the `config.pro` file.

   Note: The plotter command or system command plots all output files created in the `pro_batch_list.txt` file.

   Pro/BATCH starts processing, and creates the following files in your current working directory:
   - Object output files (<`object`>`.plt`, <`object`>`.iges`)
   - The `pro_batch_list.txt` file listing all object output files
   - The `pro_batch.log.#` file recording the results of the execution

To Run an Existing Command File

1. Type `pro_batch`. The **Pro/BATCH** dialog box opens.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select the batch file that you want to run.
4. Click **Open**. The batch file name and the objects in the batch file appear in the appropriate areas in the **Pro/BATCH** dialog box.
5. In the **Pro/Engineer Command** box, type the command.
6. Schedule the start of the batch process.

To Upgrade Files

1. Create a new command file.
2. Type the object file name.

To Create a Plot File After Creating a New Pro/BATCH Command File

1. Type the name of the object to plot.
2. Type the plotter name, for example, PostScript, or the name of the plotter configuration file to use, for example, `<filename>.pcf`. If you are using a plotter configuration file, you supply only the information not defined in the plotter configuration file.
3. Specify the required values as messages appear asking you for the following information:
   - Paper size
   - Output quality
   - User scale
   - Pen slewing
   - Page range
   - Segmented output
   - Labeled plot
4. When the message appears asking if you want to create a new plot file, type `yes` or type `no` to specify the name of an existing plot file.

   If the drawing has multiple sheets, a message appears asking if you want to create separate sheets.
5. Type `no` to append all of the sheets to a single plot file or type `yes` to create a separate plot file for each sheet of the drawing.
6. Type the name of the plot file to create.
7. At the prompt, type `yes` to export another file or type `no` to execute the command file. Pro/BATCH displays the contents of the line added to your command file.

Example: Creating and Running a Plot File of a Drawing

In the following example, Pro/BATCH creates a command file for plotting a drawing. Without exiting Pro/BATCH mode, the system immediately executes the command file. After the execution is completed, it sends the created plot file to a specified printer.

Type: `pro_batch -text <pro_command>`

```
************************************************************

********************** Batch File Execution *************

************************************************************

A carriage return after any prompt will result in the use of the value
specified within the brackets ([]). Entering the command "quit" after
any prompt will allow you to exit. If you are in command file creation,
"quit" will return you to the Batch File Execution level. Entering
"quit" at that level will get you out of the program. A list of intended
file names will be created under the name
/easter/doc/larisa/pro_batch_list.txt

************************************************************

Do you wish to run an existing command file [No]? <CR>

Enter the name for the new command file
[/easter/users/manipulator/bfile.txt]: <CR>

Using /easter/users/manipulator/bfile.txt.

************************************************************

********** Begin command file creation **********

************************************************************

Enter the type of command line to be created [1]:

1. plot

2. stl

3. iges

4. dxf

5. cadamin

6. cl_file

7. set

8. vda

9. step

10. cgm

11. vrml

12. upgrade

Enter: 1

Enter the full name of the object []: shaft.drw

Enter the plotter name [default]: <CR>

Enter the paper size [A]: <CR>

Enter desired output quality [default]: <CR>
```

```
Enter desired user scale [1.0]: 2.0

Enter the pen slewing desired.

1. No pen slew.

2. Slew

Default [1], Enter: <CR>

Enter Page range specification:

1. All Pages

2. Current Page

3. Range of Pages

Default [1], Enter: 2

If output is larger than the paper, do you want segmented output [No]:
<CR>

Do you want to label the plot [No]: <CR>

Do you want to create a new plot file [Yes]: <CR>

Do you want to create separate plot files if the drawing has multiple
sheets [No]: <CR>

Enter the Output plot file name [plot.plt]: <CR>

************************************************

The following command line has been added to
/easter/users/manipulator/bfile.txt:

-plot -object /easter/users/manipulator/shaft.drw -plotter default -
paper A -userscale 2.000000 -plotfile plot.plt

************************************************

Would you like to create another command line [No]: <CR>

************************************************

******* End of Command file Creation ********

************************************************

Do you want to execute the batch command file [Yes]? <CR>

************************************************

The list of files to be created can be found under
/easter/users/manipulator/pro_batch_list.txt.

************************************************

Do you want to plot the output file(s) ? [No]: yes

Enter the system command: laser

ss_plot_job: -plot

********* Batch File Execution Successfully Completed **********
```

To Export to STL

1. Specify the name of the model to export to STL.
2. Specify the required values as messages appear asking you for the following information:
   - Binary or ASCII output format for the stl file.
   - Output quality of the file.
   - Name for the stl file.
3. Type `yes` to export another file or `no` to execute the command file.

To Export to VRML

1. Create a file to use the Pro/BATCH option commands.

   In the **VRML Export** option, a separate line appears for each unique database.
2. Type one of the following commands:
   - For a direct export, type:

     `-vrml -object <object name> -out_path <path> -direct <y>`
   - For an in-session export, type:

     `-vrml -object <object name [-out_path <path> -direct <n>`

     where:

     `<object name>`—Indicates a part or assembly, such as `wheel_1.prt` or `engine_u.asm.2.`

     `-out_path <path>`—Indicates the path to optionally divert `.wrl` files to a specified location.

     `-direct <y or n>`—The `y` indicates a direct export and the `n` indicates an in-session export for the batch process
3. Type the following command to execute the Pro/BATCH option:

   `PROE_COMMAND 0 0 3 -batchfile [file_name]`

To Export to IGES, DXF, SET, VDA, or STEP

1. Type the name of the model to export. The contents of the additional line in the command file appears.
2. Type `yes` to export another file or type `no` to execute the command file.

   Note: If you are creating a DXF file, you can only export a drawing.

Importing Proprietary Surface with Pro/TOOLKIT

About Using Pro/TOOLKIT to Import Proprietary Surfaces

With Pro/TOOLKIT, you can import proprietary surfaces into Pro/ENGINEER. These imported surfaces are treated by the system as a special type of surface feature.

One of the possible applications of a proprietary surface is surface replacement, when solid surfaces are replaced with a Pro/TOOLKIT surface. After the proprietary surface is incorporated into the part geometry, all other geometric operations, such as Offset, Cut, and Round, are available for the surface.

Note: A model containing a proprietary surface can be retrieved, manipulated, or saved only with Pro/TOOLKIT.

Before a surface is imported, a coordinate system must be present in the part for the surface to reference.

To Import a Foreign Surface

1. In a part, click **Insert** > **Advanced** > **Blend From File** > **Surface**.
2. Select the class of the surface from the **SURF CLASS** menu. This menu lists all the classes that have been properly defined in Pro/TOOLKIT.
3. Type the name of the surface.
4. Select from the namelist menu the name of the coordinate system that the surface will reference.

If imported geometry uses dimension units different from those of the current model, those units are converted to the units of the current model.

With Pro/SURFACE, you can create nonsolid surfaces using some of the same techniques you use when you are creating solid geometry, as well as more sophisticated surface modeling methods. With Interface for Pro/ENGINEER, you can import surfaces from other CAD systems.

Working with Pro/WEB PUBLISH

About Pro/WEB PUBLISH

Pro/WEB PUBLISH enables you to:

- Create a paperless environment that reduces the cost of creating and plotting drawings.
- Extend the access to data to all parts of an organization.

The publishing tools enable you to export and then view in HTML, VRML, CGM, and JPEG formats the following on the Web browser:

- Assembly models and data

- Assembly processes including graphical and textural information for each process step
- Manufacturing processes including graphical and textural information for each process step

Process and assembly information that you view in the browser reflects the process data extracted programmatically from Pro/ENGINEER.

All publications that you create with Pro/WEB PUBLISH are view-only. After creating a publication, it is independent from the original assembly or process plan. If you make changes to the model in Pro/ENGINEER, you must recreate the publication to reflect the latest changes.

Requirements for Viewing a Publication

The requirements for viewing a publication are as follows:

- A procgi server application must be installed on your Web server. See the *Pro/ENGINEER Installation and Administration Guide* for information on individual server installation instructions.
- You can use Netscape Browser 3.0 (or higher) or Microsoft Explorer 3.0 (or higher). The browser must support HTML, JavaScript, and Java, as well as VRML, CGM, and JPEG, either inherently or with plug-ins or helper applications. The Web browser must also support frames.

Supported Graphics Formats

Pro/WEB PUBLISH supports the following graphics formats:

- **JPEG (Joint Photographic Experts Group)**—Raster-based 2D image format. This is the default format.
- **CGM (Computer Graphics Metafile)**—Vector-based 2D image format.
- **VRML (Virtual Reality Modeling Language)**—Web-based language used to describe the layout of 3D objects and environments.

To Create a WEB Publication

1. Export a Pro/ENGINEER assembly or process plan in WWW formats. The system creates a publication directory with all publication-related files.
2. Place publication directories for assemblies or process plans in the Web server's documentation root directory.
3. Use a suitable Web browser to view the publication.

Naming Conventions for Publication-Related Files and 2D Model Views

For each publication, the following files are generated:

Publication	Publication Directory and Related Files
Assembly Process	<PROCESSNAME>_PRS_WWW directory, which includes: Top-level .html files A directory for each process step, which contains .html, .vrml, .cgm, and .jpg files for that step
Assembly	<ASSEMBLYNAME>_ASM_WWW directory, which includes .html, .vrml, .cgm, and .jpg files
Manufacturing Process	<PROCESSNAME>_PRS_WWW directory, which includes: Top-level .html files A directory for each process step, which contains .html, .vrml, .cgm, and .jpg files for that step

If you want a particular assembly view to appear as the default orientation for 2D model display, create a view with the name WEB_DEFAULT. This view becomes the default view orientation for all process steps. If you have not created the WEB_DEFAULT view, the last saved orientation of the model is used as the default view with the name default.

To export step-specific views, create named views according to the following naming convention: step#_<user_defined_view>. When you are viewing the publication, the view name appears as <user_defined_view> without the step# prefix. Note that within the publication, different steps can have the same view names.

Types of Data Extracted

When you create a publication, the system extracts data from your assembly or process. The following table shows the type of data extracted for each of the publishing tools.

Publication	Type of Data
Assembly Process	Part lists 2D and 3D model representations Model parameters that you include in the assembly Model tree Step parameters, including Step #, Step type, and Step description; time estimate and cost estimate (if defined)

Assembly	Part lists 2D and 3D model representations Model parameters that you include in the assembly Model tree
Manufacturing Process	2D and 3D representations for each step showing the workcell/workpiece and the material removal (where appropriate) Step/MFG parameters (buttons to call up a browser screen to view additional information) Operations/Steps tree Model tree

To Prepare a Model for Export

1. If you want a publication to contain several JPEG or CGM orientations of an assembly, create named views in Assembly mode before you export the assembly. Otherwise, the only available CGM representation is the default view of the assembly.
2. Set configuration options to customize the Pro/WEB PUBLISH output or to configure VRML data.
   - `www_export_geometry_as`
   - `www_tree_location`
   - `www_add_aux_frame`
   - `vrml_explode_lines`
   - `vrml_background_color`
   - `vrml_file_duplicate_material`
   - `vrml_multiple_views`
   - `vrml_export_resolution`
3. To list additional model or manufacturing parameters, add them to the model before exporting the model.

Additional Model Parameters

You can specify model parameters for export in Pro/ENGINEER using the Model Tree tool. Add the model parameters to the Model Tree columns that contain the model parameters to export.

Alternatively, you can create a `conf_www.tbl` file and place it in the current working directory. In the `conf_www.tbl` file, list the parameters to include in the designated order. The following is a sample of a `conf_www.tbl` file.

```
Ref#
Qty
Name
COST
VENDOR
WEIGHT
```

To Export Process Data from Pro/ENGINEER

1. In Pro/ENGINEER, retrieve the assembly process, assembly, or manufacturing process that you want to export.
2. Configure the Model Tree to display the model parameters to export.
3. Click **File** > **Save a Copy**. The **Save a Copy** dialog box opens.
4. In the **Type** box, select **WWW**.

   The system creates a publication directory and stores all publication-related files in this directory. If the name of the publication is the same as that of an existing publication in the current directory, a message appears asking if you want to overwrite it. Pro/ENGINEER exports all user-defined views, and the default view of the assembly process, assembly, or manufacturing process.

Viewing an Assembly Process Publication

About Viewing an Assembly Process Publication

You can browse the assembly process information on the Web in a step-by-step format. Frames appear containing the following information for the current step:

- Description of the current step
- Table of parts to be assembled in the current step
- 2D or 3D graphical display of the assembly at the current step
- Model tree (optional)
- Control panel

Step Parameters

The title of the frame describing the step indicates the step number and type of step. The contents of the frame include the step description, time estimate, and cost estimate (if defined).

The step description includes the information that the process planner entered while in Pro/PROCESS for ASSEMBLIES. This can consist of procedural information (for example, a list of instructions), the purpose of the step, and any other relevant comments.

Part List

In part list frame, a table listing parts to assemble in the current step and any fixtures or tools required to perform the step, appears. By default, the part list table consists of the following items:

- Reference number
- Model name
- Quantity

The table also lists the additional parameters you include in the model when preparing the process for export. The additional information can include the part type (for example, standard component, cable, pipe, or fixture), inventory location, parameter name, and parameter value. Additional parameters that you include in the assembly process appear in the part table for all steps.

Click the name of the part in the table to open a window in which the VRML model of that part appears.

Step Navigation Control Buttons

The control buttons for navigating through the assembly process include:

- **Prev**—Show information about the previous assembly process step.
- **Next**—Show information about the next assembly process step.
- **Step 1**—A menu for selecting a step in any order appears. The name of the step appears next to the step (if you defined the name in the assembly process using **Setup/Name**).

Graphics Frame

The graphics frame shows the assembly, including parts to assemble in the current step. When you open a publication, default 2D JPG assembly view appears in the frame. Click **2D JPG** to open the window and select the graphics format you want: **2D JPG**, **2D CGM**, or **3D VRML**. The choices depend on the graphics formats selected for the export.

If your publication contains multiple 2D model views, click **View:default** and select the view you want from the list.

Depending on how the step is defined in Pro/PROCESS for ASSEMBLIES, a VRML model can show a model in its exploded state as a simplified representation, or as a full representation. You can pan, zoom, or spin a 3D model by using the VRML viewer controls.

When you drag the mouse over a VRML part, the part name appears. Click the part name in the part list or model tree, to open a window in which the VRML model of that part appears.

Typically, you can zoom or pan only a 2D CGM model. Note that 2D view manipulation tools depend on the plug-in or helper application that you use.

Viewing the Model Tree

To view a model tree, click **Tree**. Parts up to and including the current step in hierarchical order, appear with a tree in a new window. To close the window, click **Tree** again. If the tree window remains open while you move to another step, the tree updates accordingly.

By default, the tree is in a collapsed state. To expand or collapse the tree click the icon next to the assembly or subassembly name.

The number of instances of a part or subassembly within a higher-level assembly appears in parentheses next to the component name.

Viewing a Manufacturing Process Publication

About Viewing a Manufacturing Process Publication

You can browse the manufacturing process information on the Web in a step-by-step format. Frames appear containing the following information for the current step:

- Description of the current step
- Table of operations and steps for the manufacturing process
- Control panel
- 2D or 3D graphical display of the assembly at the current step

Step Parameters

The title of the frame describing the step indicates the step number and type of step. The contents of the frame include the step description, time estimate, and cost estimate (if defined). It also contains links to details about the step.

The step description reflects the information that the process planner specified while in Pro/PROCESS for MFG. This can consist of procedural information (for example, a list of instructions), the purpose of the step, and any other relevant comments.

When you select any of the following, a separate window containing information opens:

- Operation Parameters

  Click **Operation Parameters** in the step parameters frame and a separate window opens with information based on the operation parameters for the step.

- Step MFG Parameters

Click **Step MFG Parameters** in the step parameters frame and a separate window opens with information based on the step manufacturing parameters for the step.

- Tool Information

  Click **Tool Information** in the step parameters frame and a separate window opens with information based on the tool information for the step. If a tool path is associated with the step, it appears along with the tool table information.

  Note: Tool information is available only if a tool is defined for the selected step.

- WorkCell Parameters

  Click **WorkCell Parameters** in the step parameters frame and a separate window opens with information based on the work cell parameters for the step.

- NC Seq. Parameters

  When you choose **NC Seq. Parameters** from the step parameters frame, a separate browser screen opens with information based on the NC sequence parameters for the step.

  Note: NC Sequence parameters are available only if NC sequences are defined for the selected step.

- BOM Report

  Click **BOM Report** from the step parameters frame, a separate browser screen opens with the bill of materials (BOM) for the step

- Operations/Steps List

  The operations/steps frame displays an operations/steps list, showing the parts and steps in the manufacturing process. You can choose from the Operations/Steps tree to view the selected step.

Step Navigation Control Buttons

The control buttons for navigating through the manufacturing process include:

- **Prev**—Show information about the previous manufacturing process step.
- **Next**—Show information about the next manufacturing process step.
- **Step 1**—Display a pop-up menu for selecting a step in any order. The name of the step appears next to the step (if you have defined the name in the manufacturing process using Setup/Name).

Graphics Frame

The graphics frame shows the assembly, including parts to be assembled in the current step. When you open a publication, the frame displays a default 2D JPG assembly view. By clicking the **2D JPG** control button, you open the pop-up window, which enables you to select the desired graphics format: **2D JPG**, **2D CGM**, or **3D**

VRML. The choices depend on the graphics formats that were selected for the export.

If your publication contains multiple 2D model views, you can select the required view by clicking the **View:default** button and choosing a view from the list.

Depending on how the step was defined in Pro/PROCESS for Manufacturing, a VRML model can show a model in its exploded state, as a simplified representation, or as a full representation. You can pan, zoom, or spin a 3D model by using the VRML viewer controls.

For a VRML model, when you move the mouse pointer over a part, the part name appears at the mouse pointer. By clicking the part name in the part list or model tree, you can open a window displaying the VRML model of that part.

Typically, you can zoom or pan only a 2D CGM model. Note that 2D view manipulation tools depend on the plug-in or helper application that you use.

Viewing the Model Tree

To view a model tree, click the **Tree** button in the control panel. This opens a window with a tree, which displays parts up to and including the current step in hierarchical order. To close the window, click the Tree button again. If the tree window remains open while you move to another step, the tree is updated accordingly.

By default, the tree is in a collapsed state. By clicking the icon next to the assembly/subassembly name, you can expand or collapse the tree.

The number of instances of a part or subassembly within a higher-level assembly appears in parentheses next to the component name.

Viewing an Assembly Publication

About Viewing an Assembly Publication

The browser screen is divided into frames: the table of parts in that assembly, graphics window, and control buttons.

Part Table

The part table includes the following columns:

- System-generated reference number for each part
- Part name
- Total number of parts in the assembly
- Additional part parameters that you designated for export

By clicking the name of a part, you open a separate window, displaying that part in the VRML format.

Graphics Frame

When you open a publication, the graphics frame shows a default 2D JPG assembly view. By clicking the **2D JPG** control button, you open the pop-up window, which lets you select the desired graphics format: **2D JPG**, **2D CGM**, or **3D VRML**. The choices depend on the graphics formats that were selected for the export.

If your publication contains more than one 2D model view, you can select the desired view by clicking the **View:default** button and selecting a view from the list.

Depending on how the model was defined in Pro/ENGINEER, a VRML model can show a model in its exploded state, as a simplified representation, or as a full representation. You can pan, zoom, or spin your 3D model by using the VRML viewer controls. When you move the mouse pointer over a part, the part name appears at the mouse pointer. By clicking the part name, you can open a window displaying the VRML model of that part.

Typically, you can zoom, rotate (in the screen plane), or pan a 2D model. Note that 2D JPG view manipulation tools depend on the plug-in or helper application that you use.

Button Panel

The button panel contains up to three buttons, depending on the graphics formats that have been selected for export. The control buttons are:

- Button for displaying the pop-up menu to select the graphics format (2D JPG is the default option). This button is absent if only one graphics format was exported.
- Button for displaying the pop-up menu to select a 2D view to be displayed. This button is absent if only VRML was exported.
- Tree button, which opens a model tree window.

Viewing the Model Tree

To view a model tree, click the **Tree** button in the control panel. By default, the tree is in a collapsed state. By clicking the icon next to the assembly/subassembly name, you can expand or collapse the tree.

The number of instances of a part or subassembly within a higher-level assembly appears in parentheses next to the component name.

Customizing Publication Sites

About Customizing Publication Sites

In addition to customizing the Pro/WEB PUBLISH output through configuration options, the Web administrator can customize your publication sites by editing the `index.html` file in the publication directory so as to change some of the frame names or add more frames with a specific name. Once the `index.html` file is tailored

to your company requirements, the Web administrator can use that customized index.html file to set up other publication sites.

You can customize the following aspects of the publication:

- Select formats to use for model representation
- Add an additional frame to include the model tree inside the Web page
- Add additional frames, including step-specific frames

  Note: Do not remove the PRO_WEB_PANEL frame from the index.html file. Without that frame you cannot load the publication into the Web browser.

To Set Formats for Assembly Representations

If you export an assembly or assembly process data in all supported formats (that is, if the www_export_geometry_as configuration option is set to all), you can edit the index.html file to limit the model representations to the particular ones you need. To do so, edit the following line in the index.html file:

```
<"frame name="PRO_WEB_ALL" src="dummy.html"
```

and replace the frame name PRO_WEB_ALL with one of the following names:

- PRO_WEB_CGM_VRML—to view CGM and VRML
- PRO_WEB_VRML—to view VRML only
- PRO_WEB_CGM—to view CGM only
- PRO_WEB_JPG—to view JPEG only
- PRO_WEB_JPG_VRML—to view JPEG and VRML

To Include the Tree Tool in the Web Page

You can include the model tree in the main browser window by setting the www_tree_location configuration option when exporting an assembly or assembly process data. Alternatively, you can edit the index.html file and add an additional frame with the name JAVABOM as follows:

```
<"frame name="PRO_PUBLISH_JAVABOM"src="dummy.html"
```

To Add a Step-Specific Frame

You can add an additional frame to one or several process steps, and use that frame to provide any relevant information about the model.

1. Before exporting data in Web formats, set the www_add_aux_frame configuration option to yes.
2. Add the <filename>.html file to each step directory where you want to include an additional frame. The contents of the <filename>.html file can be different for each step.

3. Edit the top-level `index.html` file to reference the `<filename>.html` file by changing the frame name `PRO_WEB_AUX_step` to `PRO_WEB_AUX_<filename>`.

Adding Additional Frames

The Web administrator can add several customized frames (for example, a mail icon, title, or company logo) to an assembly process or assembly publication.

Note: When you add an additional frame to an assembly publication, place the `<filename>.html` file in the same directory as the `index.html` file.

Example: Adding a Different Video for Each Process Step

To add a different video for each process step, specify the name of the frame as `PRO_PUBLISH_AUX_VIDEO_FRAME` and add a `VIDEO_FRAME.html` file to each step directory. The contents of the `VIDEO_FRAME.html` might vary for each step.

Working with Digitized Input

About Digitized Input

Digitizing from a tablet enables you to create point, line, and arc data from existing drawings and layouts for import to Pro/ENGINEER.

Pro/ENGINEER supports the following tablets for digitizing data on an IBM RS/6000 workstation only:

- Calcomp 33240
- Calcomp 95602

To Import Data from the Digitizing Tablet

1. Create a drawing of the appropriate size.
2. Click **File** > **Open**. The **File Open** dialog box opens.
3. Select **Tablet** in the **Type** box. As you enter Digitizing mode, icons for all of the Pro/ENGINEER windows are created, and a window displays the results of the digitizing input.
4. Specify the scale and units for the imported data.
5. Calibrate the tablet so that the software can account for stretch, warpage, or tilt of the drawing being digitized.
6. Begin digitizing data after a crosshair appears to show the location of the puck on the tablet.

You can return to any one of the other modes to recalibrate or rescale the digitized data at any time.

Summary of Tablet Options

The following table summarizes all of the tablet options.

Button Reference Table

Mode	Button	Action
Scale	1-9	Specifies height and width data.
	A	Moves to the previous entry in the menu.
	B	Moves to the next entry in the menu.
	C	Toggles to next mode.
	F	Returns to Input mode.
Calibration	0	Selects point on tablet. Highlights position on calibration grid to which this is closest.
	1	Moves highlighted position up one grid point.
	3	Confirms selection of the grid point.
	5	Cancels grid point selection.
	6	Moves highlighted position right one grid point.
	9	Moves highlighted position down one grid point.
	C	Toggles to next mode.
	F	Returns to Input mode.
Input	0	Starts a chain of lines; creates a point. Use button 7 to pick existing point. When in query, used to confirm deletion.
	1	Pans up.
	2	Resets view to display whole calibrated area.
	3	Stops chain creation. When in query, used to abort process.
	4	Pans left.
	5	Centers view on screen.
	6	Pans right.

	7	Joins two chains. Selects the first chain endpoint, using 0, then the chain endpoint to connect to using 0.
	8	Starts a chain of lines. Creates a point; use button 7 to pick existing point. When in query, used to confirm deletion.
	9	Pans down.
	A	Zooms out by factor of 2.
	B	Zooms in by factor of 2.
	C	Toggles to next mode.
	D	Deletes a point. Point nearest cursor is highlighted. Upon query, use button 0 to delete, 3 to keep.
	E	Deletes a chain of line segments. Cursor must be near point on chain. Upon query, use button 0 to delete, 3 to keep.
	F	Exits tablet input. Upon query, use button 0 to confirm exit, 3 to stay in digitizing mode.

To Enter Scale Mode

1. Within Digitizing mode, press C to switch to Scale mode.
2. Specify the height and width of the drawing being digitized.
3. If you are scaling the drawing, type the scaled values.
    - Press the 0–9 buttons to specify values for height and width. Press D to specify a decimal point.
    - Press A to switch to the next option.
4. Specify the units for the drawing, and press 0 to switch between them. You can use the following values:
    - Inches
    - Millimeters (mm)
    - Abstract (scales the digitized data into the units of the current drawing)
5. Press C to exit the dialog box. You cannot exit unless all entries are valid.
6. Press F to quit Scale mode.

Scale Mode

In Scale mode, you can specify the height and width of the area being digitized, and the units of the digitized data. This input enables you to scale data, or convert it automatically to different units. For example, if the drawing on the tablet is a C-size drawing, and you want to scale it to an A-size, for height type [8.5], and for width type [11]. The digitized data is scaled down from the C-size dimensions to fit the A-size sheet. Similarly, you can scale up digitized data.

To Calibrate the Tablet

1. Place the drawing on the tablet.
2. Attach the tablet firmly to prevent it from shifting during digitizing. If it shifts, no existing data is lost, but you must calibrate the tablet again.
3. Press C to switch to Calibration mode.
4. Press A to select the calibration type.
5. Select **Simple** for linear stretch in both horizontal and vertical directions.
6. Digitize the lower-left, upper-left, and lower-right corners (the minimum needed to locate the drawing on the tablet).
7. Select **Mesh** for warpage or nonlinear stretch of the drawing.
8. Specify the number of horizontal and vertical points to calibrate.
9. Press C to show the appropriate calibration type in the dialog box. If you select **Mesh**, specify the number of points in the vertical and horizontal direction pressing the 0–9 buttons to specify values for height and width. Press D to specify a decimal point.
10. Specify the number of horizontal and vertical points to calibrate. The most simple (2 points in each direction) enables linear stretch in both horizontal and vertical directions.
11. Digitize the points for three corners of the drawing first. Press 0 to pick the point. One of the grid points that is closest to the cursor highlights. If this is the correct grid point, press 0 to confirm or 3 to cancel. If you need to select another grid point instead, move the highlight to another grid point.
12. If you select **Mesh** calibration, a grid pattern appears on the screen delineating the corners of each calibration area.
13. Press 0 to pick a calibration mark on your drawing. One of the grid points that is closest to the cursor highlights. If this is the correct grid point, press 0 to confirm or 3 to cancel. If you need to select another grid point instead, move the highlight to another grid point.
14. Continue to calibrate all areas.
15. Press C to switch to the next mode.

Calibration Mode

In Calibration mode, you can set up the tablet to create accurate input data from any source drawing. Calibrating the tablet to a drawing can compensate for warpage or tilt when the drawing is placed on the tablet. To calibrate a drawing, you can specify an area on the drawing and confirm its location on the screen. You can specify one large area (the entire drawing), for drawings that are in perfect shape, or any number of user-specified equal areas to calibrate smaller portions of the drawing to limit the effects of warpage.

To Create a Chain of Straight Line Segments

1. Press 0 to start the chain.
2. With the puck crosshairs located at the start of the line, press 0. As you move the crosshair to each new location, the next line in the chain rubberbands.
3. Continue to add points to the chain pressing 0.
4. When you have digitized the last point in the chain, press 3. The chain is examined to determine if it can be converted into a single line or arc.
5. As it highlights each segment, press 0 to convert that portion of the chain or press 3 to cancel the conversion.

   You can convert the chain in small portions, or you can continue to cancel the conversion of each segment until the entire chain is highlighted. If you accept pressing 0, the entire chain is converted into a line or arc.

Digitizing Input

All input is displayed on the screen during the digitizing process. You can pan/zoom while digitizing during all Input mode functions to easily verify the data as you create it.

Scaling capability is also available at any time. The digitized points in the tablet coordinates are maintained. Therefore, changing the scale does not alter the real coordinates of the existing digitized data.

Tip: Converting Chains Using Explicit Commands

You can also convert chains after modifying them or through explicit commands. The chain color is black if it is a polyline of straight segments, and purple if it is a segment of chain converted to an arc.

To Delete a Point

You can delete a single point or a point in a chain pressing D in the Input mode. When the point is defining a chain, the adjacent chain segments are deleted, and if it is in the middle of a chain, it breaks the chain into two.

1. Place the cursor near the point to delete and press D. The point highlights in red.
2. Press 0 to confirm deletion or press 3 to abort.

To Digitize Points

You can digitize a single point or create a chain of straight line segments.

To digitize a single point, press 0 twice, and then press 3.

To Delete a Chain

1. Place the cursor near a point on the chain to delete and press E. The chain highlights in red.
2. Press 0 to confirm deletion or press 3 to abort.

To Join Two Chains

You can join chains created at different times, or broken by deleting points in the middle of them.

1. Place the cursor on an endpoint of a chain and press 0. The point highlights and you are prompted to select another chain endpoint. To abort joining the chains, press 3.
2. Move the cursor to an endpoint of the other chain. Press 0. The two chains become one with a new line segment between the two selected endpoints.

To Convert a Chain

You can convert a chain to an arc or a single straight line pressing 8 in Input mode. This enables you to digitize a number of points to approximate an arc, or for a long line when necessary to account for warpage. Then convert the chain into exact entities.

You can convert a green chain to a line, and a purple chain to an arc.

1. Place the cursor near a point on the chain to convert, press 8.
2. From the menu, select **Arc** or **Line**.

To Configure your IBM RS/6000 to Recognize the Calcomp

1. Connect the Calcomp tablet to the IBM RS/6000 serial port.
2. Login as `root` to configure the serial port.
3. Type [`smit`] at the prompt to run the System Management Interface Tool.
4. When the SMIT window appears, select **Devices**.
5. Select **TTY** from the **Devices** listing.
6. When the TTY listing appears, check to see which devices are already defined, then modify or create a device for the Calcomp tablet. To do this, add the device. Click **Add a TTY**.
7. Select **tty rs232 Asynchronous Terminal**.

8. Select the serial port to which you connected the Calcomp tablet.
9. For the TTY configuration, make sure that these values are used:
    - **BAUD rate**—9600
    - **INPUT map file**—none
    - **OUTPUT map file**—none
    - **CODESET map file**—sbcs
    - **STATE to be configured at boot time**—available
10. When you have confirmed the above values, click **Do**. When creating a device, the system states that TTY is available with the message `tty0 is now available`.
11. Click **Exit** to exit the `SMIT` utility.
12. Type `cd /dev`
13. Type `ln -s /dev/tty# caltablet`, where `#` is the TTY number of the tablet, which appeared in step 10.
14. Check that the `getty` utility is not running. If it is, kill it by typing `pdisable tty#`, where `#` is the TTY number of the tablet.
15. Type `chmod a+rw /dev/tty#`, where `#` is the TTY number for the tablet.
16. Log out.
17. Log on as any user.

To Configure the Tablet

After the workstation is configured, you can use the bank of lights, numbers, and letters at the top of the tablet to configure it.

1. Switch on the tablet.
2. Select **Config** on the tablet using the tablet puck.
3. As you move the puck crosshairs across the letters and numbers, the power light goes on and off. When the power light is on, this represents a 1. When the power light is off, this represents 0.
4. Click **A** under the Bank options on the tablet. Configure the tablet for Bank A options using the settings shown in the following table.

Bank A Configuration

Button	1	2	3	4	5	6	7	8	9
Setting	1	1	0	1	0	1	0	0	1

Power Light	on	on	off	on	off	on	off	off	on

Button	10	11	12	13	14	15	16	17	18
Setting	1	0	1	0	1	1	1	0	1
Power Light	on	off	on	off	on	on	on	off	on

5. Click **B** under the Bank options on the tablet. Configure the tablet for Bank B options using the settings shown in the following table.

Bank B Configuration

Button	1	2	3	4	5	6	7	8	9
Setting	0	0	1	1			0	0	0
Power Light	off	off	on	on			off	off	off

Button	10	11	12	13	14	15	16	17	18
Setting	0	0	0	0	0	0	0	0	0
Power Light	off	off	off	off	off	off	off	off	off

6. When you have finished, store the configuration. If you save it to number 1, it is automatically configured when you turn on the tablet. If you store it to another number, restore the configuration before using the tablet with Pro/ENGINEER.

To Troubleshoot Problems That Occur when Digitizing

The following table describes the problems that can occur when digitizing, and their possible causes and resolution.

Description	**Possible Cause**	**Resolution**
The cursor in the digitizing window moves in the opposite direction of the puck.	The tablet has been turned off and on in Digitizing mode.	Check tablet configuration. If needed, reconfigure, then store the settings. Click in the digitizing window with the IBM mouse.

The digitized points appear incorrect.	The tablet has been turned off and on.	Restore the tablet configuration. Check tablet configuration if needed.
You are not in Digitize mode.	You already have a device other than the tablet named calcomp.	Use the `tablet_device_name` configuration option.

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