Knowledge Base

Suggested Technique for IGES Configuration Options When Transferring Data between Pro/ENGINEER and Specific CAD/CAM Applications

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Procedure

1. Choose the appropriate software application and use the following information to customize Data being transferred to/from other CAD/CAM applications. The term **Export**, as it is used in this document, refers to Exporting from Pro/ENGINEER and **Import** refers to Importing into Pro/ENGINEER:

A.) Alias | B.) ANSYS | C.) Anvil
D.) Applicon | E.) AutoCAD | F.) Auto-Trol
G.) CADDS 4.x | H.) CATIA | I.) CGS(GM)
J.) CV Personal Designer | K.) Computervision | L.) Euclid
M.) HP ME-30 | N.) HP ME-10 | O.) ICEM Surf
P.) I/EMS 2.0 | Q.) Interleaf V6.0 | R.) MISSLER
S.) PDGS | T.) SDRC I-DEAS 6.1
U.) STRIM100(V5) | V.) Unigraphics v10

MANUFACTURING SOFTWARE:

X.) CADKEY | Y.) CAmanD | Z.) CAMAX
AA.) Delcam Duct5 7.0 | BB.) MasterCAM | CC.) SmartCAM 5.0
DD.) SmatCAM 7.0 | EE.) Work NC(Nov. 93)

See Figure 1 for a Table of suggested IGES configuration options.

A.) Alias

Importing *via IGES*

Listed below are feasible ways to work with Alias data in Pro/ENGINEER.

- In Alias, remove the surface boundary trim on all surfaces, then merge the surfaces back together in Pro/ENGINEER.
- Export from Alias and read "from file" datum curves only, then create surfaces in Pro/ENGINEER.
- Redefine surface boundaries within Pro/ENGINEER using Pro/LEGACY.
- Use Alias data as a template only, and then create your own Pro/ENGINEER model.

In Alias: (Before export to Pro/ENGINEER)

1. Use Alias version 5.0.
2. Increase accuracy:

```
Curve fit distance - 0.000001
Raised surface - 0.000001
Trim cop gap tol. - 0.000001
```

3. Reduce the number of points and control curves.
4. Break all polylines at logical vertices. (ex: break a square at each corner.)
5. Break or split revolved surfaces into two parts. (Revolved surfaces can be tweaked in Alias, thus distorting its original shape. These surfaces must also be broken or split.)
6. Remove all fillets and rounds. (Alias does not trim the surfaces under these features, which creates a problem in Pro/ENGINEER.)

In Pro/ENGINEER: (Before import from Alias)

1. Create datum plane axes using offset, manually typing in "0,0,0" for coordinate system placement.
2. In certain rare cases, it may be necessary to slightly increase the accuracy of the part before import.

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

B.) ANSYS

Importing *via IGES*

In certain cases, it may be necessary to increase the accuracy of the part before exporting from Ansys.

See Figure 1 for suggested IGES configuration options.

C.) Anvil 5000

Importing *via IGES*

The default configuration settings work well for 3D wireframe information.

Exporting *via IGES*

The default configuration settings work well for 3D wireframe information.

See Figure 1 for suggested IGES configuration options.

D.) Applicon BRAVO's Gem Modeler

Exporting *via IGES*

The Gem Modeler can only accept wireframe geometry. In the EXPORT IGES dialog box, put a checkmark by **Wireframe Edges** only.

You can convert the IGES file into a GEM file by running Bravo's "IGES modeler" command at the system prompt. This command converts the `.igs` file into a `.ddd` file which can be read by GEM.

See Figure 1 for suggested IGES configuration options.

E.) AutoCAD

Importing *Drawings via IGES*

Set the Config.pro options:

```
"fix_autocad_iges_text_scale"          "yes" or "no"
"iges_zero_view_disp"                  "as_is"
```

F.) Auto-Trol

***Importing** via IGES*

Set the Config.pro option:

```
"iges_zero_view_disp"                    "as_is"
```

While this option fixes the placement and number of displayed dimensions and notes, the dimensions and notes may not be to the same scale as the geometry. Because of the differences between Pro/ENGINEER and Auto-Trol functionality, data imported from Auto-Trol is very unpredictable.

G.) CADDS 4.x

***Exporting** via IGES*

See Figure 1 for suggested IGES configuration options.

H.) CATIA

***Importing** via IGES*

Before exporting from CATIA, set the following parameter:

```
"CATMOD"         "8" or less   (15 is default)
```

***Exporting** via IGES*

Refer to the Suggested Technique for CATIA Data Transfer

When not using Pro/CAT, the surfaces may not appear trimmed in CATIA. This is because CATIA creates two layers which contain trimmed and untrimmed surfaces.
In CATIA, place all surfs in **NO SHOW**, then **SHOW, ALL FACES (SCATTERED LINES)**, then **GRAPHICS MODE, SHOW, BOUNDARIES OF FACES**.

See Figure 1 for suggested IGES configuration options.

***Importing** Drawings via IGES*

Set the Config.pro option:

```
"fix_catia_iges_sym_note"                "yes"
```

***Exporting** Drawings via IGES*

Set the Config.pro options:

```
"iges_out_catia_gdt_width"               "yes"
"iges_out_catia_notes"                   "yes"
"iges_out_symbol_entity"                 "no"
```

I.) CGS (Internal General Motors Software Package)

***Importing** via IGES*

Set the Config.pro options:

```
"intf_in_treat_polyline_as"              "single_spline"
"intf_in_blanked_entities"               "yes"
```

In certain cases, it may be necessary to increase the accuracy of the part before importing from CGS.

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

J.) CV Personal Designer

Importing *via IGES*

Set the Config.pro option:

```
"fix_imported_set_view_orient"        "yes"
```

K.) Computervision

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

L.) Euclid

Importing

- From 3D Solid module - export from Euclid using SET, with no Boolean csG (use Facet BREP approximation)
- From 3D Surface module - IGES out from Euclid with default option (i.e., no splines, but Bezier output in Euclid option).
- From 2D Draft Master module - No information is currently available.

Exporting

- To 3D Solid module - transfer not possible since Euclid uses Boolean csG with possible Facet BREP approximation.
- To 3D Surface module in Euclid v. 3.0 - See Figure 1 for suggested IGES configuration options. (It reads Bezier patches.)
- To 2D Draft Master module - Exporting SET files yields the best result from Pro/ENGINEER.

M.) HP ME-30

Importing *via IGES*

The default configuration settings are recommended.

Exporting

ME-30 is not capable of importing data.

N.) HP ME-10 (2D Drawings)

Importing *Drawings via IGES and DXF*

2D data from HP ME-10 translates well into Pro/ENGINEER on SGI hardware with default settings.

***Exporting** via IGES*

2D data translates well from Pro/ENGINEER on SGI hardware with the default settings.

O.) ICEM Surf

***Exporting** via IGES*

See Figure 1 for suggested IGES configuration options.

VDA will successfully transfer 3D data to ICEM Surf.

P.) I/EMS 2.0

***Importing** via IGES*

Use default IGES options when exporting from EMS.

Set the Config.pro option:

```
"intf3d_in_include_items"              "srfs_crvs_pnts"
```

***Exporting** via IGES*

I/EMS will not accept IGES surface entities 128, 141, or 143.

See Figure 1 for suggested IGES configuration options.

Q.) INTERLEAF V 6.0

Exporting

The default config.pro options work well for CGM, IGES drawings, HPGL, TIFF, and Postscript output. For CGM output, however, Interleaf leaves some Pro/ENGINEER lines white on a white background. Changing the color of the part before export will not solve the problem. Instead, redefine the system colors in Pro/ENGINEER by selecting **Colors**, **System** from the **Utilities** pull down menu. Select the desired color scheme from the **Scheme** pull down menu.

R.) MISSLER (VISIMIL V 1.21)

***Exporting** via IGES*

See Figure 1 for suggested IGES configuration options.

S.) PDGS

***Importing** Parts and Assemblies via IGES*

For more information regarding the Pro/DATA_for_PDGS option, refer to the Suggested Technique for Transferring Data from PDGS into Pro/ENGINEER .

Turn NURBS on. (The system administrator may have to do this.) From the Data Collection Menu, Function 14 Convert, Turn On B-Splines, Exit. Select Part, Set Default to Master Picture, Surfaces - convert to NURBS (all), some may have been created as NURBS. IGES OUT, Flavors IGES 5.0.

Exporting *Parts and Assemblies via IGES*

For more information regarding the Pro/DATA_for_PDGS option, refer to the Suggested Technique for Transferring Data from Pro/ENGINEER into PDGS .

See Figure 1 for suggested IGES configuration options. The system administrator for the PDGS system may need to "turn NURBS on" in order to interpret all of the surfaces in the IGES file.

When exporting assemblies to PDGS, create a layer for each part. Give each layer an ID # by selecting **Layer**, **Setup layer**, **From file**, **Mod params**. Edit the list of layers and assign a number from 1 to 254 to each layer. This ID # will correspond to a PDGS picture named "Level #" when the IGES file is read into PDGS. Place each part on its layer and export.

Exporting *Drawings via IGES*

Turn off all datums, curves, etc. you do not want included in the IGES file.
Set "text_width_factor" to "1.0" in the drawing setup file.

T.) SDRC I-DEAS 6.1 *Exporting* *via IGES*

See Figure 1 for suggested IGES configuration options. This will provide an IGES file which should import into I-DEAS with the default I-DEAS settings. If I-DEAS has difficulties trimming any surfaces, the underlying surface will be brought into I-DEAS but the trimming curves will be imported as wireframe curves (instead of edges). If there is a problem, it is usually on a fillet surface. Most of the time these faces can be repaired within I-DEAS by recreating them with the surface-by-boundary technique (pick the edges from the neighboring faces). Loosening up the tolerance during the IGES import into I-DEAS generally allows the IGES files to import without any errors (all trimmed surfaces successfully mapped to faces). This may eliminate the need to rebuild any faces. If there are problems reading an IGES file directly into the I-DEAS Geometry module, another option is to try reading it into the FEM module and then transferring the part to the geometry module.

I-DEAS is sensitive to the accuracy of the model. It wants an IGES "minimum resolution" of 0.01 mm. This value is shown in the `igesout.log` file created by Pro/ENGINEER when exporting geometry. In certain cases, it may be necessary to increase the accuracy of the part before export.

U.) STRIM100 (V 5)

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

V.) Unigraphics V. 10

Importing *via IGES*

The default settings provide good results. If possible, IGES files from UG should be created such that the surfaces are described with entity type 128.

Importing *Drawings via IGES*

Set the Config.pro options:

```
"iges_zero_view_display"                "as_is"
"iges_note_display"                     "as_is"
```

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

MANUFACTURING SOFTWARE

X.) CADKEY

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

Y.) CAmanD

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options. Create the CAmanD job file from the IGES file using CAMAX's IGES to CAmanD translator.

Z.) CAMAX

Importing *via IGES*

When importing Pro/ENGINEER IGES files into CAMAX, better results may be achieved using CAMAX's command line IGES translator.

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

AA.) Delcam Duct5 7.0

Delcam Duct5 (Release 7) supports IGES Version 4.0.

Within Delcam Duct there exists an 'IGES In' and 'DDX' option to read in IGES files. Pro/ENGINEER files read in correctly and quicker with the 'DDX' command.

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

BB.) MasterCAM

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

CC.) SmartCAM 5.0

Exporting

1. Set the Config.pro options:

```
"default_draw_scale"                    "1"
"put_iges_drawing_entity"               "yes"
"use_iges_font_1003"                    "no"
"use_iges_kanji_font_2001"              "yes"
"iges_out_mil_d_28000"                  "no"
"iges_out_start_note"                   "no"
"iges_out_catia_gdt_width"              "yes"
"iges_out_catia_notes"                  "no"
"iges_out_dwg_line_font"                "yes"
"iges_out_dwg_pnt_ent"                  "no"
"iges_out_symbol_entity"                "yes"
```

2. Create a temporary drawing. (Do not use the standard drawing because it would include information not required for cutter path creation, i.e. title block, notes, etc.) Select drawing size.
3. Select **Views** (Select the model name to open)
4. Select **Add view**, **General**, **Full view**, **No Xsec**, **No scale**, **Done**.
5. Select a location for the model view.
6. Orient the part to the desired view using named views or orientation commands.
7. Select **OK**, **Done return**
8. Select **Utilities**, **Environment** (Unselect the following options: **Datum Planes**, **Point Symbols**, **Point Tags**, **Datum Axes**, **Coordinate Systems**)
9. Select **Apply**, **OK**.
10. Select **File**, **Save As**, and set the Type to **IGES** (Enter output file name, not exceeding 8 character DOS limitation.)

DD.) SmartCAM 7.0

Exporting *via IGES*

Set the Config.pro option:

```
"iges_out_assembly_default_mode"        "flat"
```

See Figure 1 for suggested IGES configuration options.

For Version 7.0, use either 2D method, as above for version 5.0, or 3D option as in the following example:

1. Retrieve the part model.
2. Create a coordinate system with the desired orientation on the part model.
3. Select **Utilities**, **Environment** (Unselect the following options: **Datum Planes**, **Point Symbols**, **Point Tags**, **Datum Axes**, **Coordinate Systems**)
4. Select **File**, **Save As**, and set the Type to **IGES** (Enter output file name, not exceeding 8 character DOS limitation.) Select **Wireframe Edges** (Select desired coordinate system), **OK**.

EE.) WorkNC (Nov. 93)

Exporting *via IGES*

See Figure 1 for suggested IGES configuration options.

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2. The following settings are suggested based on empirical testing done between Pro/ENGINEER and the named

application. These options are subject to change and may not apply to all available versions of the application mentioned. Therefore, these options should only be used as a guide for transferring data. Figure 1 is a summary of suggested configuration option settings for IGES-specific export of 3D geometry.

The IGES configuration options in Figure 1 can be modified in the IGS_Config.pro file by selecting "Options" from the Export IGES dialog box, or by adding the options to Config.pro.

Figure 1: Suggested Config.pro Settings For Exporting 3D Geometry from Pro/ENGINEER to Specific CAD/CAM Applications

Pro/ENGINEER 20.0	**iges_out_spl_crvs_as_126**	**iges_out_all_srfs_as**	**iges_out_spl_srfs_as_128**	**iges_out_trim_xyz**	**iges_out_trim_curve_deviation**	**iges_out_trm_srfs_as_143**	**iges_out_mil_d_28000**	**intf3d_out_extend_surface**	**intf_out_blanked_entities**
Default	yes	default	yes	yes	curr.accu.	no	no	yes	yes
Alias	*	128	*	no	-	-	-	-	-
ANSYS	*	128	*	*	-	*	*	-	no
Anvil 5000	*	*	*	*	-	*	*	-	*
Applicon Bravo	no	-	-	-	-	-	-	-	-
CADDS (4.X)	*	128	*	*	-	*	*	-	no
CADKEY	no	128	*	*	-	*	yes	-	no
CAMAND	*	128	-	*	-	-	-	-	-
CAMAX	*	default /128	*	*	-	-	-	-	-
CATIA	*	128	*	*	-	*	*	-	no
CGS (Internal GM)	*	*	*	*	-	*	*	-	*
Computer-Vision	-	-	-	*	-	-	-	-	-
Delcam Duct5 7.0	yes	128	yes	yes	-	no	no	no	no
Euclid	no	114	no	no	-	*	*	-	-
ICEM Surf	*	128	*	*	-	*	*	*	no
I-EMS 2.0	*	*	no	*	-	*	-	-	-
Master-CAM	*	128	*	*	-	*	*	-	no
MISSLER (VISIMIL V.1.21)	*	128	*	*	-	*	*	-	no
PDGS	*	128	*	*	-	*	*	-	no
SDRC I-DEAS 6.1	*	128	*	*	-	*	-	-	-

SmartCAM v.7.0	*	128	*	no	-	yes	*	-	-
STRIM100 (V 5)	*	128	*	no	-	*	*	-	no
UG V. 10	*	128	*	*	-	*	*	-	no
WorkNC	*	128	*	no	-	yes	*	-	-

- **Asterisks indicate the default value is preferred.**
- **No information is available for dashed entries in the table.**
