CAD Interfaces

SILMA®
Division of Adept Technology, Inc.

Table of Contents

CADDS Interface

Release Notes

Current Version	V 3.3
Module	CADDS_INTERFACE_3.3
Platform	HP, Sun, Solaris, SGI5

Introduction

The CADDS interface is a two-way translator: data transfer can be performed in two ways:

- CADDS part files can be translated into SILMA software model files.
- SILMA software models can be translated into CADDS part files.

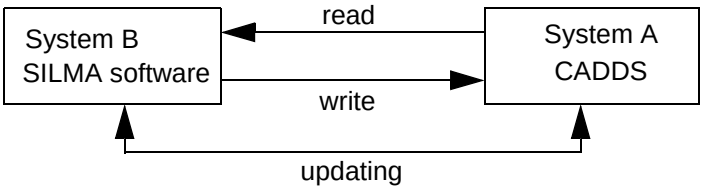
Both these operations can be performed on any platform the interface is supported. It is not required to have CADDS running in order to perform any of these operations.

The CADDS Translator user interface is a user-friendly, menu-driven interface. “Using the CAD Interface Panel” on page 4 provides a detailed description of how to use the SILMA software menus and panels.

In the SILMA software CADDS interface, there are three major classes of geometric models that can be directly translated from Computervision CADDS4X and CADDS5X to SILMA software:

- Three-dimensional wireframe models
- Three-dimensional surface models
- Three-dimensional solid models

Using this interface, the SILMA software program reads the System A data (CADDS), translates it into System B format (SILMA software), and writes it into the System B database. System B can then translate and write data back to System A and provide updates between the systems.



You can model parts in CADDS and simulate them in SILMA software—all transparently. Your CAD parts geometry is automatically updated by a menu-driven converter. A part may consist of any kind of CADDS geometry and converted to NURBS geometry at the time of translation into SILMA software.

The SILMA software CADDS interface reads, writes, and updates CADDS part models and SILMA software models. Use the interface to import CADDS models into SILMA software and to export certain geometry and data generated from simulation.

The interface lets you create CADDS parts (without even getting into CADDS) by selecting the SILMA software geometry and assembly layout. You can then export these parts to the CADDS system. The interface also interacts with the CADDS runtime process to keep the translated parts up-to-date.

Installing the CADDS Converter

If the CADDS converter is present during SILMA software installation, the **newuser** script prints a message asking you to logout and login as root to run **install_cvdors**. The **newuser** script then creates the links and starts the daemons on your system (ODB-DAEMON if the assembly interface is installed).

Several environmental variables are set as part of running the **newuser** script at the SILMA software installation. The variables required for **cvdors_login** are appended to the user's **.login** file, and **cvdorsrc** is copied to your user area and renamed **.cvdorsrc**. The **uenv** is copied to your user area and renamed **.uenv**.

Global variables used in the CADDS converter are documented in **cadds_default.sil** which exists in your product when you create a product with the CADDS converter.

CADDS4X is not needed to convert CADDS parts to SILMA software models. However, CADDS4X (including Assembly Design) is needed if you want to convert Assembly Design models.

Using the CAD Interface Panel

The CAD Interface panel is displayed by selecting *Import/Export Model...* from the **Modeling** pulldown menu.

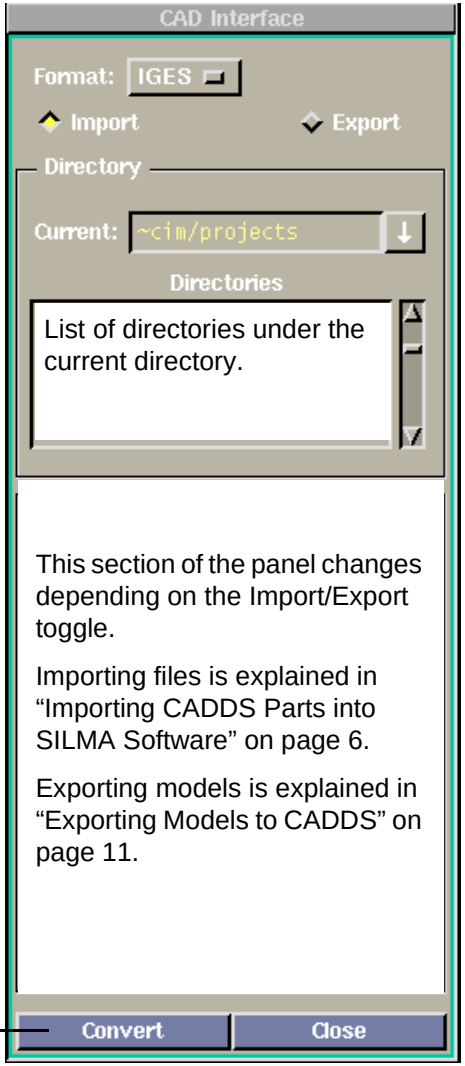


Figure 1 The CAD Interface panel

The CAD Interface panel selections are described in Table 10.

Selection	Description
Format	Specifies the format of the file that will be imported into SILMA software. The default is *Previous State*. Selecting *CADDS* enables you to translate CADDS files into SILMA software.
◆ Import	Toggle this selection to import CADDS models into SILMA software. See "Importing CADDS Parts into SILMA Software" on page 6.
◆ Export	Toggle this selection to write out a CADDS part directly from SILMA software with selected geometry. See "Exporting Models to CADDS" on page 11.
Directory	Sets the path to the directory containing the CADDS export model files. You can use the list button to select **`~cim/projects`** or **`~cim/users`**, or use the field to enter any directory on the system. The Directories list provides a list of directories under the current directory. Directories that are **not** libraries have two dots (..) after the directory name. Libraries do not have this symbol. **NOTE** A **library** is a directory that stores data and information about SILMA software entities. Selecting a directory from the Directories list makes it the current directory.
Convert	Converts the export files in the conversion queue.
Close	Cancels the settings and dismisses the panel.

Table 1 CAD Interface selections

Importing CADDS Parts into SILMA Software

The following steps illustrate how to import CADDS parts into SILMA software:

1. Display the CAD Interface panel by selecting *Import/Export Model…* from the **Modeling** pulldown menu.
2. Choose *CADDS* as the format for import using the Format choice button.
3. Toggle ◆ Import.
4. Toggle ◆ Part.
5. Set the path to the directory in which the export file resides by using the Current selection and choose the file or files to be imported by selecting the file or files from the list.
6. Switch on ❑ Add to World if desired. Switching this on installs the converted part into the current workcell.
7. Switch on ❑ Save to File if desired.

   If this is switched off, the converted part will be not be saved. Switching this on specifies that you would like to save the converted part. Either the ❑ Use From or Save Dir selections are used to specify the directory where the converted part will be saved:

   Save Dir selections are used to specify a directory where the converted part will be saved. These selections are only available if you have specified that you want to save the file by switching on ❑ Save to File and that you do not want to save the file in the current directory by switching off ❑ Use From.

   Clicking this button displays a list used to select a directory. You can also enter a directory name in the field.

   Switching on ❑ Use From saves the imported file in the current directory. Switching this off enables you to specify a different directory using the Save Dir selections. This selection is only available if you have specified that you want to save the file by switching on ❑ Save to File.

8. Click *Edit Settings…* to display the Edit Settings panel and change the settings as desired. The Edit Settings panel is described in "Editing the Settings" on page 7.
9. Click *Convert* to start the conversion process.

Importing CADDS Assemblies into SILMA Software

CADDS assemblies can also be brought into SILMA software. The procedure is basically the same as described above. The main difference is that you toggle ◆ Assembly instead of ◆ Part before selecting the files to be converted (Step 4). Toggling ◆ Assembly displays any **_db** file under the selected directory tree. The rest of the procedure is the same as described above.

Editing the Settings

The Edit Settings panel is used to edit the conversion settings when importing CADDS parts or assemblies. Clicking *Edit Settings…* in the CAD Interface panel displays the Edit Settings panel, which is shown on the following page.

Editing Resolution

Toggling ◆ Resolution enables you to modify segment length or facet size on parametric models.

Low resolutions (fewer facets) will speed up the graphics for large CAD models, however the models will look "rough". High resolution will make the models look more real, but may slow the graphics of some platforms. Setting the proper resolution is important as it can achieve a good balance between performance and appearance.

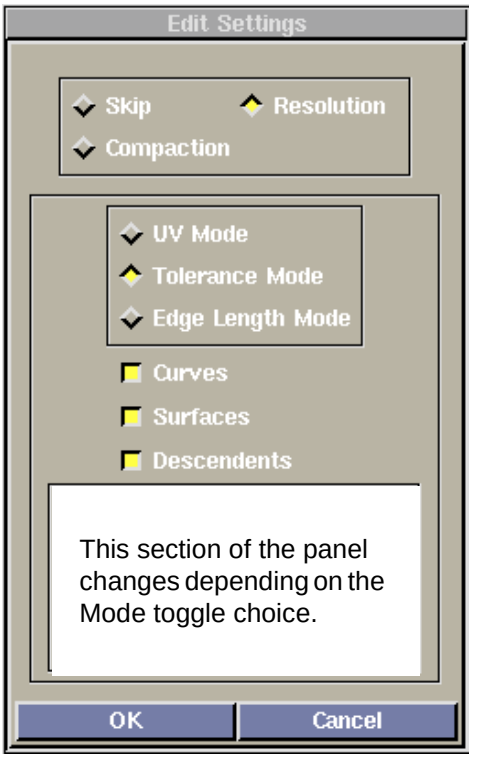


Figure 2 The Edit Settings panel (with ◆ Resolution toggled)

Table 11 describes the three modes.

Selection	Description
◆ UV Mode	Specifies the facetization as a two-dimensional grid. This is not recommended since it does not optimize the number of facets. You need to specify three fields: Surface U, Surface V and Boundary. Boundary is the resolution of the trimmed surface boundary.

Table 2 Modes

Selection	Description
◆ Tolerance Mode	Specifies the maximum allowable deviation of the faceted representation from the actual analytic surface. This optimizes the number of facets by creating more where the curvature is high, and fewer where surfaces are flat.
◆ Edge Length Mode	Specifies the size of the facets, but does not optimize.

Table 2 Modes (continued)

Table 3 describes the available switches that determine the kind of geometry that will be edited.

Selection	Description
❑ Curves	If this is switched on, resolution on curves will be edited according to the current settings.
❑ Surfaces	If this is switched on, resolution on surfaces will be edited according to the current settings.
❑ Descendents	If this is switched on, resolution on all the descendents of the selected shape will be edited according to the current settings.

Table 3 Geometry switches

Skipping Attributes

Toggling ◆ Skip displays four switches, which serve as filters:

❑ Remove Curves

If this is switched on, both discrete and analytic curves will be skipped during conversion.

❑ Remove Surfaces

If this is switched on, both discrete and analytic surfaces will be skipped during conversion.

❑ Remove Analytic

If this is switched on, the analytic description of the entities will be removed after their conversion.

❑ Remove Non-geometry

If this is switched on, entities with no geometry will be skipped during conversion.

Editing Compaction

Toggling ◆ Compaction provides selections for changing the tree structure of a specific model by creating a bond between objects. Compaction can be performed at two levels: at the top node and at the child level.

Figure 3 ◆ Compaction selections

◆ Rigid

Structures the tree of the selected object such that the child objects are considered to be affixed to the parent object, and are no longer considered separate entities. If the parent object moves, its children will move also. In addition, the children are not permitted to move relative to one another. You must move the parent to move the children.

Using rigid compaction increases performance on certain platforms.

◆ Non-rigid

Structures the tree of the selected object such that the child objects are considered to be affixed to the parent object, but the child objects can still move relative to one another.

◆ Permanent Rigid

Structures the object tree such that the all objects, whether parent or child, are considered to be on one level. This option provides maximum graphics performance and minimum memory consumption, but cannot be reversed.

Exporting Models to CADDS

This section describes how to export SILMA software models into CADDS parts files.

1. Display the CAD Interface panel by selecting *Import/Export Model…* from the **Modeling** pulldown menu.
2. Choose *CADDS* as the format for export using the *Format* choice button.
3. Toggle ◆ Export.
4. Set the path to the directory in which the export file resides.
5. Select the model to be exported from either the *Graphics Window* or the *Quick Pick Window*. Refer to “Supported Entities” on page 12 for the list of export entities supported.
6. Set the filename by double clicking on the text in the File field and entering the new filename. The default is the path of the selected model.
7. Click *Convert* to start the conversion process.

Supported Entities

CADDS to SILMA Software (Importing)

Table 5 describes the supported CADDS entities.

Entity	Type Number	SILMA software Entity
point	2	point
line	3	rbspline
arc	5	rbspline
conics (302-ellipse, 303-hyperbola, 304-parabola)	6	rbspline
n_spline	12	rbspline
n_surface	14	rbsurf
tabulated cylinder	16	rbsurf
surface of revolution	17	rbsurf
ruled surface	18	rbsurf
plane	88	

Table 4 Supported CADDS entities

Table 5 describes the supported miscellaneous **n_splines**.

Entity	Type Number	SILMA software Entity
spline (old spline type)	7	rbspline
B-spline	8	rbspline
curve pole (Cpole)	9	rbspline

Table 5 Supported miscellaneous **n_splines**

Table 6 describes the supported miscellaneous **n_surfaces**.

Entity	Type Number	SILMA software Entity
B-spline surface (Bsurface)	19	rbsurf
surface pole (Spole)	20	rbsurf

Table 6 Supported miscellaneous **n_surfaces**

Table 7 describes the supported dimensions and tolerance entities that are not part of CV-DORS.

Entity	Type Number	SILMA software Entity
feature control symbol	32	gsymbol
linear/ordinate dimension	33	ldimention
angular dimension	34	adimention
radial dimension	35	rdimention
general label	36	glabel
diameter dimension	37	ddimention

Table 7 Supported dimensions and tolerance entities

Table 8 describes the supported miscellaneous entities.

Entity	Type Number	SILMA software Entity
curve point	10	rctcurve
Vector	22	opoint
Subfigure instance	80	frame
solid or trimmed surface	91	face
face	92	trimsurf

Table 8 Supported miscellaneous entities

SILMA Software to CADDS (Exporting)

CS Entity Type	CADDS Entity Type
pcurve	nspline
dcurve	lines
single_trace	nspline
dual_trace	nsurface

Table 9 SILMA software to CADDS (write)

Catia Interface

Release Notes

Current Version	V 2.3.3
Module	CATIA_INTERFACE_2.3.3
Platform	RS/6000, HP, SGI5

Introduction

The Catia® interface provides two ways to transfer data between Catia and SILMA software:

- Catia export files can be translated into SILMA software data.
- SILMA software data can be translated into Catia export files. The files generated are of ASCII workstation format.

Both these operations can be performed on any platform on which the interface is supported. It is not required to have Catia running in order to generate Catia export files.

The interface has been built to read and install Catia models and CATEXP files generated by Catia Version 4 Release 1.5 and earlier Catia V3Rx.

The interface uses the Open Data Exchange/CAT Toolkit to read and write Catia model and project-related information. The entities supported by this interface are described in “Supported Entities” on page 27. Any entities not listed in this section are not supported.

The interface has the capability to read export files (generated by the CATEXP utility) and convert them into SILMA software **`.model`** files. This interface handles files generated by both mainframe (EBCDIC format) and RS/6000 (ASCII format).

Catia export interface writes SILMA software model data into a Catia export file. The Catia write interface writes the selected shape tree as a list of details under the root node. In other words, the shape tree is replaced by the flat list of details.

The Catia Translator user interface is a user-friendly, menu-driven interface. “Using the CAD Interface Panel” below provides a detailed description of how to use SILMA software menus and panels.

By default, Catia model files with the **`.exp`** extension are recognized as Catia files. If a different file extension is desired, you can change the file extension for the current state by modifying the **`'<file_extension>'`** field:

IMPORTANT	Do not change **`file_extension`** to **`model`**. SILMA software files use the **`.model`** extension, so setting the **`catia_file_extension`** global to **`model`** causes ambiguity. You should also note that the Catia interface does not have the ability to read Catia-generated **`.model`** files (even if they are renamed).

NOTE	Text commands must be entered at the SIL> prompt. You can use the SIL> prompt in a shell window by selecting *Exit Menus* from the **File** pulldown menu. To obtain access to the menus and panels, enter **menus();** at the SIL> prompt when you are finished entering text commands.

Using the CAD Interface Panel

The CAD Interface panel is displayed by selecting *Import/Export Model...* from the **Modeling** pulldown menu.

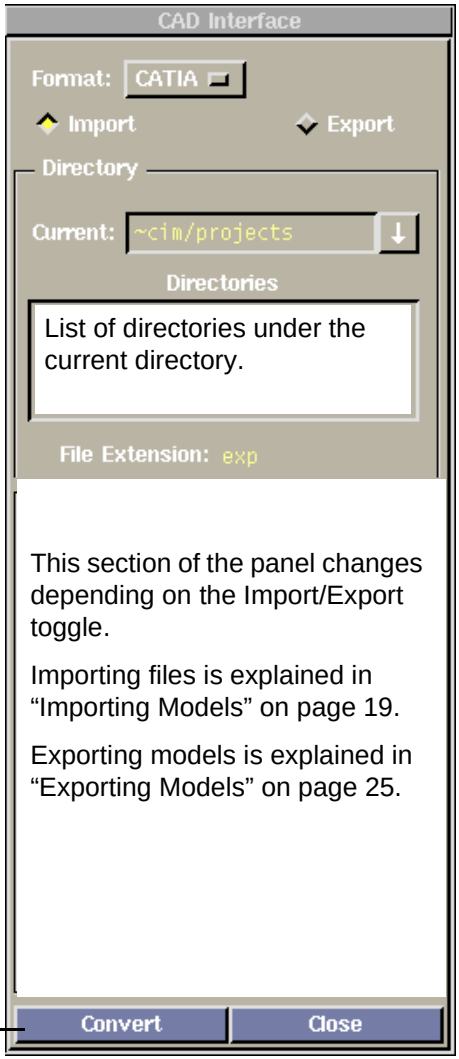


Figure 4 The CAD Interface panel

The CAD Interface panel selections for the Catia interface are described in Table 10.

Selection	Description
Format	Specifies the format of the file that will be imported into SILMA software. The default is *Previous State*. Selecting *CATIA* enables you to translate Catia files into SILMA software.
◆ Import	Toggle this selection to import Catia models into SILMA software. See "Importing Models" on page 19.
◆ Export	Toggle this selection to write out a Catia part directly from SILMA software with selected geometry. See "Exporting Models" on page 25.
Directory	Sets the path to the directory containing the Catia export model files. You can use the list button to select **`~cim/projects`** or **`~cim/users`**, or use the field to enter any directory on the system. The Directories list provides a list of directories under the current directory. Directories that are **not** libraries have two dots (..) after the directory name. Libraries do not have this symbol. **NOTE** A **library** is a directory that stores data and information about SILMA software entities. Selecting a directory from the Directories list makes it the current directory.
File Extension	Specifies the file extension that will be used to identify Catia files. **IMPORTANT** This field should not be set to model. SILMA software files use the **`.model`** extension, so setting the File Extension to model causes ambiguity.
Convert	Converts the export files in the conversion queue.
Close	Cancels the settings and dismisses the panel.

Table 10 CAD Interface selections

Importing Models

To import a Catia model into SILMA software, follow these steps:

1. Display the CAD Interface panel by selecting *Import/Export Model…* from the **Modeling** pulldown menu.
2. Choose *CATIA* as the format for import using the *Format* choice button.
3. Toggle ◆ Import.
4. Set the path to the directory in which the export file resides by using the Current selection and choose the file or files to be imported by selecting the file or files from the list.
5. Switch on ❑ Add to World if desired. Switching this on installs the converted model into the current workcell.
6. Switch on ❑ Save to File if desired.

   If this is switched off, the converted model will be not be saved. Switching this on specifies that you would like to save the converted model. Either the ❑ Use From or Save Dir selections are used to specify the directory where the converted model will be saved:

   Save Dir selections are used to specify a directory where the converted model will be saved. These selections are only available if you have specified that you want to save the file by switching on ❑ Save to File and that you do not want to save the file in the current directory by switching off ❑ Use From.

   Clicking this button displays a list used to select a directory. You can also enter a directory name in the field.

   Switching on ❑ Use From saves the imported file in the current directory. Switching this off enables you to specify a different directory using the Save Dir selections. This selection is only available if you have specified that you want to save the file by switching on ❑ Save to File.
7. Click *Edit Settings…* to display the Edit Settings panel and change the settings as desired. The Edit Settings panel is described in “Editing the Settings” on page 21.

8. Click *Convert* to start the conversion process.

NOTE	During conversion, the interface first checks for a corresponding model file in the directory. If a current model file is found, the interface installs the model instead of converting the export file. If the model is not current (i.e., if the export file has been modified after the model file was generated), the interface updates the model by performing the conversion.

Importing Large Export Files

When export files consist of a large number of models of considerable size, SILMA software might not have enough memory to process all of them. In such cases, selective extraction of models is preferred.

Two globals are introduced to facilitate the selective extraction of models from an export file. These are

model_lower specifies which model to start at (inclusive)

model_upper specifies which model to stop at (inclusive)

Example

If you wanted to extract the 12th model only, then you would set the globals as follows:

```
model_lower := 12;      { start at 12th model }
model_upper := 12;      { stop at 12th model }
```

Similarly, if you wanted to extract all the first 7 models, the globals should be set as

```
model_lower := 1;       { start at the first model }
model_upper := 7;       { stop at the 7th model }
```

NOTE	Text commands must be entered at the SIL> prompt. You can use the SIL> prompt in a shell window by selecting *Exit Menus* from the **File** pulldown menu. To obtain access to the menus and panels, enter **menus();** at the SIL> prompt when you are finished entering text commands.

The globals are reset every time the state is started. They can be manually reset by calling the procedure

```
reset_model_limits();
```

This sets the globals to default, which is to convert all the models.

If you do not know the number of models in the export file, invoking the conversion with the default settings prints out the number of models.

Editing the Settings

The Edit Settings panel is displayed by clicking *Edit Settings...* in the CAD Interface panel.

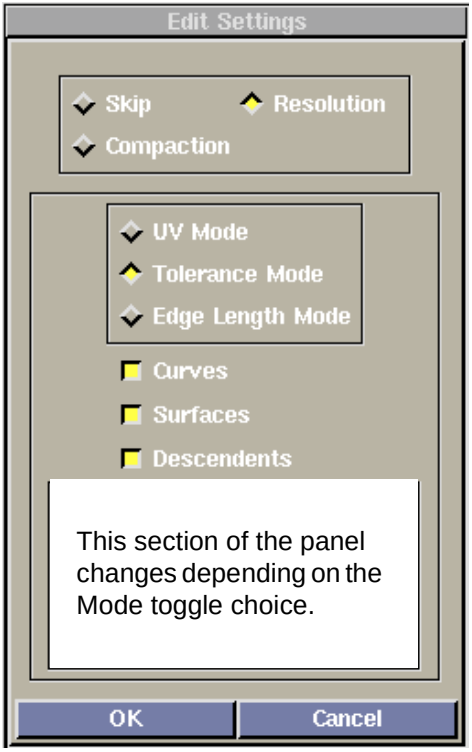


Figure 5 The Edit Settings panel (with ◆ Resolution toggled)

Editing Resolution

Toggling ◆ Resolution enables you to modify segment length or facet size on parametric models.

Low resolutions (fewer facets) will speed up the graphics for large CAD models, however the models will look “rough”. High resolution will make the models look more real, but may slow the graphics of some platforms. Setting the proper resolution is important as it can achieve a good balance between performance and appearance.

Table 11 describes the three modes.

Selection	Description
◆ UV Mode	Specifies the facetization as a two-dimensional grid. This is not recommended since it does not optimize the number of facets. You need to specify three fields: Surface U, Surface V and Boundary. Boundary is the resolution of the trimmed surface boundary.
◆ Tolerance Mode	Specifies the maximum allowable deviation of the faceted representation from the actual analytic surface. This optimizes the number of facets by creating more where the curvature is high, and fewer where surfaces are flat.
◆ Edge Length Mode	Specifies the size of the facets, but does not optimize.

Table 11 Modes

Table 3 describes the available switches that determine the kind of geometry that will be edited.

Selection	Description
❑ Curves	If this is switched on, resolution on curves will be edited according to the current settings.
❑ Surfaces	If this is switched on, resolution on surfaces will be edited according to the current settings.
❑ Descendents	If this is switched on, resolution on all the descendents of the selected shape will be edited according to the current settings.

Table 12 Geometry switches

Skipping Attributes

Toggling ◆ Skip displays four switches, which serve as filters:

❑ Remove Curves

If this is switched on, both discrete and analytic curves will be skipped during conversion.

❑ Remove Surfaces

If this is switched on, both discrete and analytic surfaces will be skipped during conversion.

❑ Remove Analytic

If this is switched on, the analytic description of the entities will be removed after their conversion.

❑ Remove Non-geometry

If this is switched on, entities with no geometry will be skipped during conversion.

Editing Compaction

Toggling ◆ Compaction provides selections for changing the tree structure of a specific model by creating a bond between objects. Compaction can be performed at two levels: at the top node and at the child level.

Figure 6 ◆ Compaction selections

◆ Rigid

Structures the tree of the selected object such that the child objects are considered to be affixed to the parent object, and are no longer considered separate entities. If the parent object moves, its children will move also. In addition, the children are not permitted to move relative to one another. You must move the parent to move the children.

Using rigid compaction increases performance on certain platforms.

◆ Non-rigid

Structures the tree of the selected object such that the child objects are considered to be affixed to the parent object, but the child objects can still move relative to one another.

◆ Permanent Rigid

Structures the object tree such that the all objects, whether parent or child, are considered to be on one level. This option provides maximum graphics performance and minimum memory consumption, but cannot be reversed.

Exporting Models

Toggling ◆ Export provides selections for converting SILMA software models to Catia Export files.

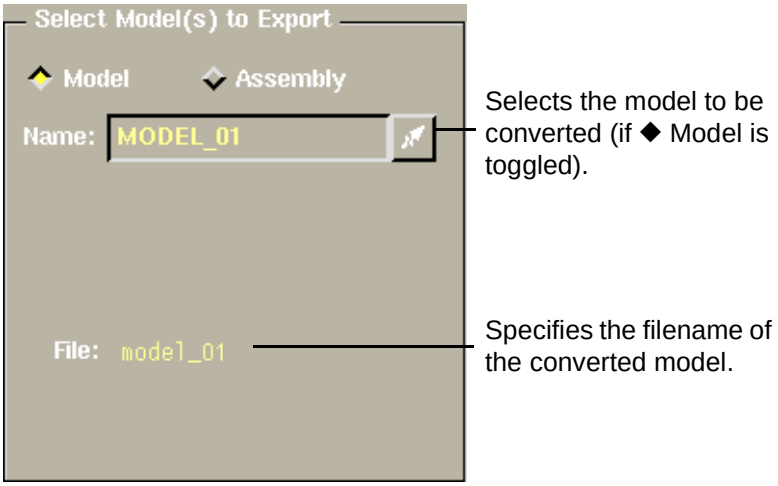


Figure 7 ◆ Export selections

The Catia Export interface writes back a Catia Version 3 export file. The Catia export files generated by the Export interface can be read back into SILMA software using the Catia import interface. Only one model at a time can be written out into an export file.

Wireframe data is converted into cache segments and stored as a bunch of line segments. Connected curves are stored as composite curves and disconnected curves are stored as individual line segments. Solids/surfaces are written out as polyhedral solid/surface data.

Stripped surfaces (no analytical information) cannot be written into Catia. As the entities written into Catia export file are discrete, their resolution cannot be altered after exporting. In order to achieve the

desired resolution, the shape tree passed into the export interface should be pre-processed to that resolution. We recommend you keep the resolution within reasonable limits because the file size grows proportionately.

To export a SILMA software model into Catia, follow these steps:

1. Select *Import/Export Model…* from the **Modeling** pulldown menu.
2. Choose *CATIA* as the format of export using the *Format* choice button.
3. Toggle ◆ Export.
4. Set the path to the directory in which the export file is written.
5. Select the model to be exported from either the *Graphics Window* or the *Quick Pick Window*. Refer to “Supported Entities” on page 27 for the list of export entities supported.
6. Set the filename by double clicking on the text in the File field and entering the new filename. The default is the path of the selected model.
7. Click *Convert* to start the conversion process.
8. The system will prompt you at the end of the process.

NOTE	The Catia interface does not support mirrored entities and dittos.

Trouble Shooting

This section describes some of the file-related problems.

ERROR	Incompatible file format.
SOLUTION	This error can be caused by one of the following: ■ The export file being read-in does not conform to the Catia export file formats. Check if the file is a valid Catia export file generated by either the Catia export utility or the SILMA software-Catia interface. ■ There may be unwanted control characters in the file. Check if the file is compressed/coded.

ERROR	Error in writing export file.
SOLUTION	This error occurs while exporting a model into Catia. It can be caused by one of the following: ■ The user does not have permission to write in the directory the file is being written into. ■ There is not enough space in the directory.

Supported Entities

Table 13 describes the space (3D) entities supported.

Entity	Type Number	SILMA software Entity
point	1	point shape

Table 13 Space (3D) entities supported

Entity	Type Number	SILMA software Entity
line	2	rctcurve
parametric curve	3	rbspline
plane	4	plsurf
polynomial surface	5	rbsurf
trimmed surface	6	trimsurf
volume	7	trimsurf
3-axis system	8	frame
transformation	9	frame
edge	12	surfcurve
skin	13 (as 6)	trimsurf
polyhedral surface	16	facet set
ditto	28	copy of entity with frame
composite curve	24	pcurvelist
Conics:		
circle, ellipse, parabola, hyperbola	20, 21, 22, 23	circle, ellipse, parabola, hyperbola
NURBS curve	46 (as 3)	rbspline
NURBS surface	47 (as 5)	rbsurf

Table 13 Space (3D) entities supported (continued)

Table 14 describes the solid entities supported.

Entity	Type Number	SILMA software Entity
polyhedral solid	17	facet set
exact solid	17	trimsurf

Table 14 Solid entities supported

SLA Interface

Release Notes

Current Version	V 2.0
Module	SLA_INTERFACE_2.0
Platform	HP700, RS/6000, SGI5, Sun, Solaris

Introduction

The SLA interface provides two ways to transfer data between SLA and SILMA software:

- SLA data files can be translated into SILMA software.
- SILMA software data can be translated into SLA-triangularized **`.stl`** format files.

Both operations can be performed on any platforms that support the SLA interface.

The SLA interface imports SLA files. SLA data is brought across as discrete facets. Since SLA writes out only facet data, the facets do not contain any analytical information. The model's resolution is determined by the resolution at which the model was written out into the SLA file. Once imported, the model's resolution cannot be edited.

Using the CAD Interface Panel

The CAD Interface panel is displayed by selecting *Import/Export Model...* from the **Modeling** pulldown menu.

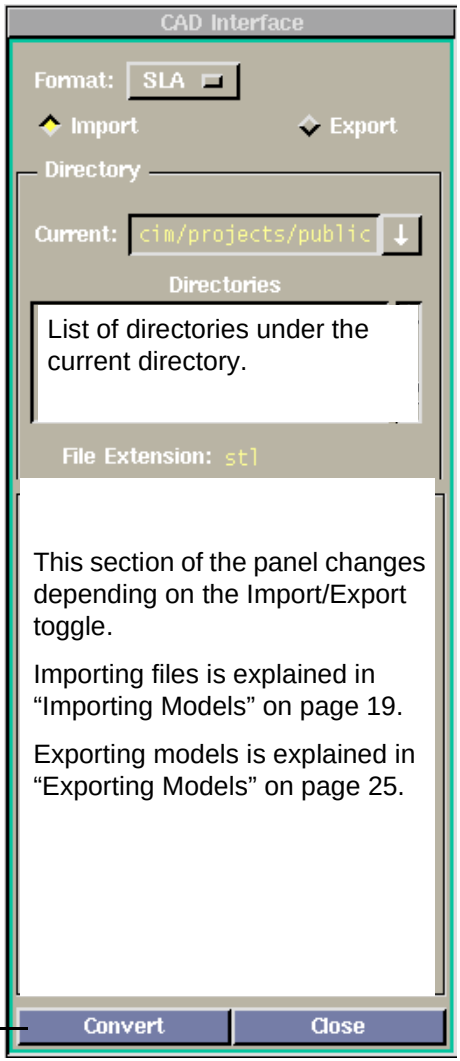


Figure 8 The CAD Interface panel

The CAD Interface panel selections for the SLA interface are described in Table 10.

Selection	Description
Format	Specifies the format of the file that will be imported into SILMA software. The default is *Previous State*. Selecting *SLA* enables you to translate **`.stl`** files into SILMA software.
◆ Import	Toggle this selection to import SLA models into SILMA software.
◆ Export	Toggle this selection to write out a **`.stl`** part directly from SILMA software with selected geometry. See "Exporting Models" on page 25.
Directory	Sets the path to the directory containing the SLA export model files. You can use the list button to select **`~cim/projects`** or **`~cim/users`**, or use the field to enter any directory on the system. The Directories list provides a list of directories under the current directory. Directories that are **not** libraries have two dots (..) after the directory name. Libraries do not have this symbol. **NOTE** A **library** is a directory that stores data and information about SILMA software entities. Selecting a directory from the Directories list makes it the current directory.
File Extension	Specifies the file extension that will be used to identify SLA files.
Convert	Converts the export files in the conversion queue.
Close	Cancels the settings and dismisses the panel.

Table 15 CAD Interface selections

Importing Models

To import a SLA model into SILMA software, follow these steps:

1. Display the CAD Interface panel by selecting *Import/Export Model…* from the **Modeling** pulldown menu.
2. Choose *SLA* as the format for import using the *Format* choice button.
3. Toggle ◆ Import.
4. Set the path to the directory in which the export file resides by using the Current selection and choose the file or files to be imported by selecting the file or files from the list.
5. Switch on ❑ Add to World if desired. Switching this on installs the converted model into the current workcell.
6. Switch on ❑ Save to File if desired.

   If this is switched off, the converted model will be not be saved. Switching this on specifies that you would like to save the converted model. Either the ❑ Use From or Save Dir selections are used to specify the directory where the converted model will be saved:

   Save Dir selections are used to specify a directory where the converted model will be saved. These selections are only available if you have specified that you want to save the file by switching on ❑ Save to File and that you do not want to save the file in the current directory by switching off ❑ Use From.

   Clicking this button displays a list used to select a directory. You can also enter a directory name in the field.

   Switching on ❑ Use From saves the imported file in the current directory. Switching this off enables you to specify a different directory using the Save Dir selections. This selection is only available if you have specified that you want to save the file by switching on ❑ Save to File.

7. Click *Convert* to start the conversion process.

NOTE	During conversion, the interface first checks for a corresponding model file in the directory. If a current model file is found, the interface installs the model instead of converting the export file. If the model is not current (i.e., if the export file has been modified after the model file was generated), the interface updates the model by performing the conversion.

Exporting Models

Toggling ◆ provides selections for converting SILMA software models to SLA Export files.

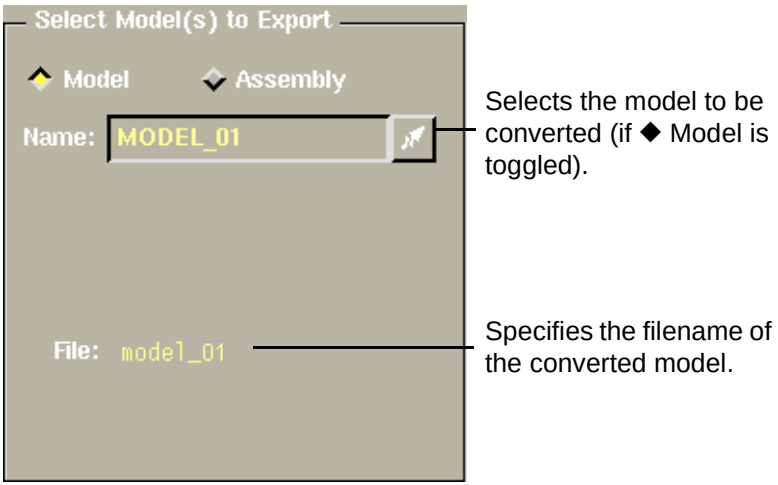


Figure 9 ◆ Export selections

The export interface looks at all the nodes with cache (display data) and writes them into the **`.stl`** file. Blockc's are directly written out and meshes are converted into blockc before they are written. Linesets (wireframe data) are not supported by SLA.

All geometric entities are exported from SILMA to SLA. The only except is wireframe data, since SLA does not support it.

Notes on Working with SLA Files

- Blockc's are expected to have convex facets. If they do not, set "**force_convex_facets**" to **TRUE** (which is the default) and re-create their cache. Failure to do this may result in the facets being incorrect after generation.
- The exported shape will be created with respect to its current pose in the current units. Note that SLA files do not store units. Set the current units to the desired units and, if necessary, re-attach the pose onto the object before exporting the shape.
- As entities written out into the **`.stl`** file are discrete, their resolution cannot be altered after exporting. In order to achieve the desired resolution in the receiving system, the shape tree passed into the export interface should be pre-processed to the desired resolution. Because the file size grows proportionately, you should keep the resolution within reasonable limits.

Pro/ENGINEER Interface

Release Notes

Current Version	V 3.0
Pro/ENGINEER Version	Rel 17
Platform	RS/6000, HP, SGI 5.3, Solaris 5.5

The Pro/ENGINEER® to SILMA software interface converts Pro/ENGINEER solid model parts and assemblies into SILMA software models and workcells, respectively. This interface requires the Pro/ENGINEER software to be running along with SILMA software for the data transfer to be accomplished.

Pro/ENGINEER parts are converted into SILMA software and saved as **`.model`** files. During conversion of Pro/ENGINEER assemblies, individual parts are stored as **`.model`** files and the final assembly is stored as a cell in SILMA software. Any part or assembly that has been previously converted is not re-converted.

The interface has been built to transfer Pro/ENGINEER solid model part data into the SILMA software. The entities supported by this interface are listed in “Supported Entities” on page 40. The Pro/ENGINEER part to be transferred should be a solid-model part.

The Pro/ENGINEER interface enables you to bring across geometric tolerance (GD&T) information along with the geometry of the part or assembly. GD&T data attached to a part are extracted and stored in their respective classes. All the GD&T associated with the part are stored under an empty node named GDT

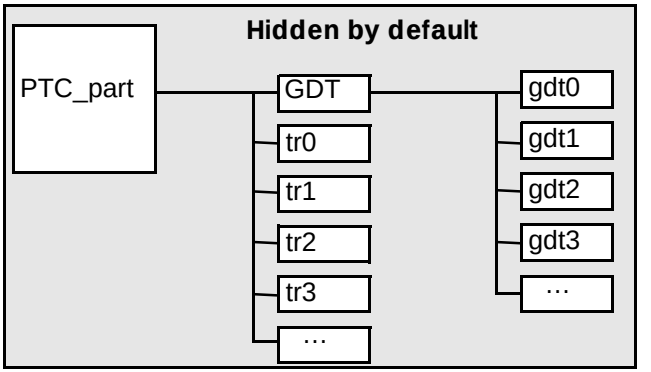
under the main part node. GDT node is at the same level as the geometry. By default, all the GD&T information is made invisible. Users who need visual access to the information can see it by using the “unhide” panel selections.

NOTE

CimStation Inspection Users:

GD&T information can be used in CimStation Inspection products for direct code generation. See the “Tolerances” section on page 8-13 of the *CimStation Inspection User’s Manual* for more details.

The following illustration depicts the tree structure of an extracted part that has been imported from Pro/ENGINEER into SILMA software.



PTC_part: part from Pro/ENGINEER (empty node)

GDT: empty node that holds GD&T information

tr*: trim surfaces making the part geometry

gdt*: GD&T information associated with ‘part’

If, for instance, gdt0 is perpendicularity, then its class list will be **list(emptyshape, ptc_gdt, perpendicularity, geomtol, shape)**.

The perpendicularity class contains all the necessary information about the tolerance like the **tol** value, datum or datums associated with it, qualifier, etc.

ptc_gdt class will provide the ID of the entity to which the tolerance is associated.

NOTE

Once displayed, the GD&T information is view dependent. The leaders make sense only when viewed normal to the GDT box. This is the camera angle at which the part was viewed in Pro/ENGINEER during the export operation. Since some of the boxes might be enclosed inside the object, it is best to view them in wireframe mode.

GD&T associated with dimensions are displayed along the side of the part in a stack. This is due to the fact that they do not have leaders associated with them.

NOTE Datum Reference Planes are not brought across in the current release.

Getting Started

1. To start SILMA software, enter the following at the UNIX prompt:

   `sspa` or **`./sspa`**

   The Product Administration panel is displayed.

2. The Pro/ENGINEER to SILMA software interface template is named **`proe`**. Select this product by clicking on proe in the Select Product to Start list. Pro/ENGINEER

3. Click *Start Product.*

4. You will be prompted for the command to start the Pro/ENGINEER software:

```
Enter command to start Pro/ENGINEER [return to cancel]:
```

   Enter **`pro`** (or whichever command is used to start Pro/ENGINEER).

   Pro/ENGINEER will automatically start the **`proe`** template and link SILMA software to Pro/ENGINEER.

NOTE If the **`proe`** script uses change directory (CD) to start Pro/ENGINEER from a specific area, SILMA software will not be linked.

To configure the SILMA software side of the converter, you may need to edit the **`cim_ptc_config/proe_defaults.sil`** file. This file contains all the default settings for the model file being written. The defaults are set to reasonable values and may not need any modification.

Exporting Models

Following these steps will write out and save a SILMA software model file into the SILMA software project directory:

1. Start the template following the instructions outlined in the above section.
2. Create a Pro/ENGINEER solid model part or activate a Pro/ENGINEER part by selecting the part: *MODE//PART//RETRIEVE/partname.*
3. Export the part into a SILMA software model file by selecting *PART//INTERFACES//EXPORT/<SILMA software product>*.

The SILMA software project directory into which the model file is written can be changed by selecting the menu *MISC//CHANGE_SIL_DIR*. You will be prompted to enter the new SILMA software export directory. The **`lib_link`** will only be updated if the selected project directory exists. If the selected directory does not exist, the previous value of **`lib_link`** is retained.

A default directory can be set during start-up for the session by modifying the string **`lib_link`** in the **`cim_ptc_config/proe_defaults.sil`** file in the template area:

```
lib_link := '/project/demo';
```

NOTE Text commands must be entered at the SIL> prompt. You can use the SIL> prompt in a shell window by selecting *Exit Menus* from the **File** pulldown menu. To obtain access to the menus and panels, enter **menus();** at the SIL> prompt when you are finished entering text commands.

This command changes the save directory to **`/project/demo/models`** for **`.model`** files and **`/project/demo/cells`** for **`.cell`** files.

The converted model or cell file can be installed into any SILMA software state.

Trouble-Shooting

ERROR	Problems occurring while translating large assemblies.
SOLUTION	Translating large assemblies may require extra swap space. In most cases, if the system halts due to lack of swap space, the problem can be overcome by simply re-starting the state, and then the conversion process. If this procedure does not correct the problem, you might have to individually convert sub-assemblies of relatively smaller size and assemble them in SILMA software.

ERROR	Change Silma Directory Error: Directory not found.
SOLUTION	■ Check that the directory specified is not a fictitious directory. ■ Check that the directory specified is not a project directory (i.e., does not have **`models`** and **`cells`** directories in it) and that you have permission to create directories.

ERROR	Error in writing SILMA software model file.
SOLUTION	■ Check if the default directory specified as **`lib_link`** in the file **`cim_ptc_config/proe_defaults`** is a valid project directory. ■ Check that you have permission to write to the specified project directory. ■ Check that there is enough space in the directory.

Supported Entities

Faces (Trimmed Surfaces)

Pro/ENGINEER surface types:

- plane
- cylinder
- cone
- torus
- surface of revolution
- tabulated cylinder
- ruled surface
- coons patch
- fillet surface
- spline surface
- b-spline surface
- cylindrical spline surface

Edges (Trimmed Surfaces Boundary Edges)

Pro/ENGINEER curve types:

- line
- arc
- spline
- b-spline

Supported GD&T Entities

- angularity
- perpendicularity
- flatness
- line
- circularity
- parallelism
- cylindricity
- circular runout
- position
- surface
- concentricity
- total runout
- straightness

VSA Interface

Release Notes

Current Version	V 1.0
Pro/ENGINEER Version	Pro15
Module	VSA_INTERFACE_1.0
Platform	SGI 5, HP 700

The VSA interface is a one-way translator. The interface exports Pro/ENGINEER parts into SILMA software model files. Unlike the Pro/ENGINEER interface, this interface has the capability to transfer Pro/ENGINEER solid model data along with GD&T and related information into a SILMA software model file. The VSA Interface is directly integrated into Pro/ENGINEER and will appear as a menu item in the VSA-GDT menus.

This interface needs Pro/ENGINEER to be running for the data transfer to be accomplished. The interface has been built to transfer Pro/ENGINEER V15 solid model part and GD&T data into SILMA software model files. The entities supported by this interface are listed in the section “Supported Entities” on page 40. Any entities not listed are not yet supported.

The VSA Interface uses the VSA-GDT/Pro host interface to build and access functional feature models. Refer to VSA-GDT/Pro manuals for more details about the construction of a functional feature model.

The default directory can be changed during a session by selecting *ChangeSilDir* from the **Misc** main menu. You will be prompted to enter the new SILMA software export directory. The **lib_link** will only be updated if the selected project directory exists. If the selected directory does not exist, the previous value of **lib_link** is retained.

A default directory can be set during start-up for the session by modifying the string **lib_link** in the **cim_ptc_config/proe_defaults.sil** file in the template area:

```
lib_link := '/project/demo';
```

NOTE	Text commands must be entered at the SIL> prompt. You can use the SIL> prompt in a shell window by selecting *Exit Menus* from the **File** pulldown menu. To obtain access to the menus and panels, enter **menus();** at the SIL> prompt when you are finished entering text commands.

This command changes the save directory to **/project/demo/models** for **.model** files and **/project/demo/cells** for **.cell** files.

The converted model or cell file can be installed into any SILMA software state.

Getting Started

1. To start SILMA software, enter the following at the UNIX prompt:

   sspa or **./sspa**

   The Product Administration panel is displayed.

2. The Pro/ENGINEER to SILMA software interface product is named **vsa**. Select this product by clicking on vsa in the Select Product to Start list.

3. Click *Start Product.*

4. You will be prompted for the command to start the Pro/ENGINEER software:

```
Enter command to start Pro/ENGINEER [return to cancel]:
```

   Enter **`pro`** (or whichever command is used to start Pro/ENGINEER).

   Pro/ENGINEER will automatically start the **`vsa`** template and link SILMA software to Pro/ENGINEER.

NOTE	If the **`vsa`** script uses change directory (CD) to start Pro/ENGINEER from a specific area, SILMA software will not be linked.

To configure the SILMA software side of the converter, you may need to edit the **`cim_ptc_config/proe_defaults.sil`** file. This file contains all the default settings for the model file being written. The defaults are set to reasonable values and may not need any modification.

Exporting Models

1. Start the template by following the instructions outlined in the preceding section.
2. Create a Pro/ENGINEER solid model part or activate an existing one by retrieving it from memory. This can be done by selecting the menu: MODE//PART//RETRIEVE/partname.
3. Attach tolerance values on the part. This can be done by using the menu: MODE//PART//SET_UP//GEOM_TOL.

NOTE	Refer to the Pro/ENGINEER documentation for additional information.

4. Invoke VSA-GDT by selecting the button: MODE//VSA-GDT.

This menu button appears only when a part is present in the session.

5. Create a Functional Feature model out of the current part.

NOTE	Refer to the VSA-GDT manual for information on creating functional feature models and attaching tolerances to them.

6. Once all the information has been fed and edited, export the solid model part into SILMA software by selecting the menu choice: VSA-GDT//EXPORT SILMA.

The SILMA software model file is created and the model file is written in the specified SILMA software export directory.

The new model file can now be loaded into the VSA-Inspection product for further analysis.

Trouble Shooting

ERROR	Tolerance/Feature not exported.
SOLUTION	Check if the tolerance/feature has been recognized by VSA/GDT and built into the functional feature model. Note that Pro/ENGINEER features have NO relation to the functional features created by VSA/GDT. All the features and tolerances exported are listed in the *Transcript Window* during conversion.

ERROR	Change Silma Directory Error: Directory not found.
SOLUTION	■ Check that the directory specified is not a fictitious directory. ■ Check that the directory specified is not a project directory (i.e., does not have **`models`** and **`cells`** directories in it) and that you have permission to create directories.

ERROR	Error in writing SILMA software model file.
SOLUTION	■ Check if the default directory specified as **`lib_link`** in the file **`cim_ptc_config/proe_defaults`** is a valid project directory. ■ Check that you have permission to write to the specified project directory. ■ Check that there is enough space in the directory.

Appendix

Importing Large Models

Large CAD models need extra memory to be accommodated in a SILMA software session. When importing large model files or CAD models, there may be a need to increase memory. This appendix explains how to do this.

The variables that need to be modified are **alloca_stack_size** and **malloc_max**. For new products, these variables are located in the file **`cim/templates/base/lisp.ini`**

For existing products, these variables are located in the file **`cim/templates/<template name>/lisp.ini`**

If the error message **too many objects in putobj—need to increase alloca_stack_size** is displayed at any time, follow these steps:

1. At the end of the appropriate file, add the command
   alloca_stack_size = 24000000;

   or

   alloca_stack_size = 24M;
2. Add **malloc_max** to the file (it can be added anywhere) by adding the command
   malloc_max = 30000000;

   or

   malloc_max = 30M;

By increasing the **alloca_stack_size** from 6000000 to 24000000, you increase the swap space requirement by 18 MB.

NOTE Refer to the *Installation Guide* for information on swap space requirements for your specific hardware platform.