



## **Cadra Release Bulletin**

**Version 10.4**

**DM-CA-109-05**

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## Overview

This Release Bulletin is divided into seven sections, as shown in the table below. The Cadra documents in which new Version 10.4 information will appear also are noted:

Bulletin Section	Cadra Documents
Installation	<i>Cadra for Windows Installation Guide</i> <i>Cadra Installation Guide</i> HTML file on your Cadra software CD: <i>install.htm</i>
Cadra Design Drafting	<i>Cadra Reference Guide: Cadra User Interface</i> <i>Cadra Reference Guide: Windows User Interface</i> <i>Cadra Reference Guide: Entity Construction and Editing</i>
Autogeometry	<i>Cadra Autogeometry User Guide</i>
Translators: ADT and CDT	<i>Cadra Translators User Guide</i>
Options: Raster	<i>Cadra Options User Guide</i>
Cadra NC	<i>Cadra NC Reference Guide</i>
CadraWorks	<i>CadraWorks Integration Guide</i>

The new features and enhancements for Cadra Version 10.4 and related option software are outlined below.

When a new feature requires additional descriptions, a page (“Pg”) number indicates the page in this document on which detailed information begins. “N/A” indicates that the table description is complete without further detail.

Installation		
Feature	Description	Pg
Install.htm	A new installation procedure for Release 10.4 is described in an HTML file, <i>install.htm</i> , which is included with your software. Print and review this file, close your browser, and follow the installation steps.	N/A

<b>Installation (continued)</b>		
<b>Feature</b>	<b>Description</b>	<b>Pg</b>
Cadra.ini Variable to Inhibit Dimension Creation	A new variable in the cadra.ini file enables you to inhibit the creation of dimensions in Draw Mode.	8
Cadra.ini Variable for CadraWorks	A new variable in the cadra.ini file enables you to access both SolidWorks drafting and Cadra drafting.	8
Cadra.ini Variable for DRT Units Scale	A new variable in the cadra.ini file enables you to indicate a units scale factor in a DRT file external reference.	9
NEC PC 98 Keyboard: NFER Key Assignment	You can assign the NFER key on the NEC PC 98xx series keyboard. This requires a modification to the cadra.ini file and use of Autogeometry, as described in this Bulletin.	9
Ctrl-C	The <i>Cadra for Windows Installation Guide</i> , page 38, states that Ctrl-C and Control Break are used to interrupt processing. In the Windows 95 and NT environments, Ctrl-C invokes the Copy function.	N/A

<b>Cadra Design Drafting</b>		
<b>Feature</b>	<b>Description</b>	<b>Pg</b>
Cadra.ini Variable to Inhibit Dimension Creation	As described for Cadra installation, a new variable in the cadra.ini file enables you to inhibit the creation of dimensions in Draw Mode.	8
Autosave	The Autosave feature automatically saves your current drawing at a user-specified time interval.	10
Customizable Toolbars	(Cadra for Windows only) You can move various pushbuttons on the toolbar to suit your preferences.	11
New Colors for Workplanes	Three new system colors related to Cadra NC have been added.	11
New Color for Invalid Dimensions	A new system color related to CadraWorks has been added.	13

<b>Cadra Design Drafting (continued)</b>		
<b>Feature</b>	<b>Description</b>	<b>Pg</b>
Pen Scaling and Plot Scale Inhibit	You can indicate whether or not pens will scale with the plot if the plot scale is greater than 1.0. In addition, you can control the plot scale for system line fonts and user fonts.	13
User-Specified Plotter Escape Sequences	(Cadra for Windows only) For certain plotters, you can specify an escape sequence from the plotter setup dialog box.	15
Figure Color Display	This feature provides flexibility in assigning colors to inserted figures.	16
Surface Display	You can specify how surfaces are drawn by either using the number of display curves specified in each surface entity, or by using the same number of display curves for all surface entities.	19
NURB to Bezier Spline Conversion	The EXPLDE NRB option available with the GROUP function is used to convert a NURB spline to a Bezier spline. Bezier is the type of spline created with SPLINE, INSERT.	N/A
Object Linking and Embedding (OLE) Container	(Cadra for Windows only) Windows Object Linking and Embedding (OLE) container is supported.	23
Cadra Database Support of OLE	(Cadra for Windows only) Cadra version 10.2 and subsequent releases support OLE, and versions 10.0 and 10.1 do not.	23

<b>Autogeometry</b>			
<b>Feature</b>	<b>Description</b>	<b>Pg</b>	
PLANESURF Structure	A new Autogeometry structure named PLANESURF is added. It corresponds to the 3-D plane surface geometry in the Cadra database.	31	
New Functions	New functions are available to read, modify, and write 3-D plane surfaces; to evaluate 2-D and 3-D NURB splines; and to read and modify entity names.	32	
	READPSS\$	Reads a 3-D plane surface geometry from the database.	33
	MODPSS\$	Modifies a 3-D plane surface geometry in the database.	35
	WRITPSS\$	Creates a new 3-D plane surface entity in the database.	37
	EVALNSS\$	Evaluates a point on a 2-D NURB spline at a given parameter value.	39
	EVALN3\$	Evaluates a 3-D point on a 3-D NURB spline at a given parameter value.	41
	READST\$	Returns the name status of an entity whose name can be modified with a MODNMS\$ (TYPE = 4) subroutine call.	43
	MODNMS\$	Modifies the name of a view, figure, figure corresponding to a figure instance, or entity in the database.	45

<b>Translators: CDT and DXF</b>		
Feature	Description	Pg
CDT: Figure Option	You can now key in a Cadra internal figure name into which the CDT part will be read.	48
DXF: Construction Line Translation Option	An option is available to determine if construction lines are enabled or disabled during translation.  With the Windows User Interface, the ENABLED and DISABLED options are available on the Translator Options window.  With the Cadra User Interface, the option is a toggle on the menu.	N/A
DXF/DWG Entity Support	New read support for AutoCAD Release 14 entities has been added.	48
Spline and NURB Transfer into Cadra NC	Refer to the description of spline and NURB transfer into Cadra NC.	58

<b>Options: Cadra 3D and Raster</b>		
Feature	Description	Pg
Cadra 3D: Joining NURBs	The NURB MENU option under SPLINE offers an additional option, JOIN NURB, to join NURB splines that meet end-to-end, all lie in the same plane or are all true 3-D entities, and are tangent at the joined endpoints.	N/A
Cadra 3D: Work Plane List	The Work Plane List dialog (or submenu for the Cadra User Interface) shows saved work planes and enables you to select, delete, or name them.	53
Raster: Pivot Point Definition	You can specify a pivot point about which edit operations occur.	53
Raster: Rotation Angle Definition	Rather than selecting from predefined rotation angles, you can key in any angular value.	54

<b>Options: Cadra 3D and Raster (continued)</b>		
Feature	Description	Pg
Raster: Resolution Control	You can specify a resolution value to use when rasterizing vector entities.	54

<b>Cadra NC</b>		
Feature	Description	Pg
Updated and Redesigned Cadra NC	All dialogs have been updated and redesigned to fit on a low resolution screen, use consistent capitalization, and provide consistent tab order. In addition, shortcuts have been added to all pull-down menus.	N/A
Cadra NC Defaults File: prf_nc.sys	You can set default options for Cadra NC from the prf_nc.sys file. This defaults file is similar to the prf.sys file used to set options in Cadra.	55
Function Key Mappings	The mappings for several function keys have changed.	58
New Colors for Workplanes	As described for Cadra Design Drafting, three new system colors related to Cadra NC have been added.	11
Spline and NURB Transfer into Cadra NC	Splines and NURBs created in several CAD products can be transferred into Cadra NC through a series of conversions.	58
Improved Error Handling	Informative error messages are displayed which are linked by a function key to more detailed information. In addition, all errors that occur in the current NC session are written to a log file.	63
New or Modified Functions and Options	Several Cadra NC functions and options are either new or modified for Versions 10.4.	67

<b>CadraWorks</b>		
<b>Feature</b>	<b>Description</b>	<b>Pg</b>
Cadra.ini Variable	As described for Cadra installation, a new variable in the cadra.ini file enables you to access both SolidWorks drafting and Cadra drafting.	8
New Color for Invalid Dimensions	As described for Cadra Design Drafting, a new system color related to CadraWorks has been added.	13
Exploded View Enhancements	The feature to generate an explode view of a solid assembly in CadraWorks has been modified.	70
Cadra View of Current SolidWorks View	You can create a view in Cadra of the current SolidWorks view.	71

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## Installation

### Cadra.ini Variable to Inhibit Dimension Creation in Draw Mode

A new variable in the cadra.ini file enables you to inhibit the creation of dimensions in Draw Mode. The variable is:

`DISABLE_DRAW_MODE_DIMENSIONS`

If the variable is not defined or is set to zero (the default), the user will be able to define dimensions in Draw Mode. If this variable is defined and set equal to 1, the user will be forced into View Mode when an attempt is made to enter the DIMENSION function while in Draw Mode.

### Cadra.ini Variable for CadraWorks

A new variable in the cadra.ini file enables you to access both SolidWorks drafting and Cadra drafting. The variable is:

`ENABLE_SOLIDWORKS_DRAWINGS`

With the addition of this variable, CadraWorks can be run in one of two modes.

**Mode 1: The variable `ENABLE_SOLIDWORKS_DRAWINGS` is set to `TRUE`.**

When you create a new drawing in SolidWorks (open the FILE menu and select NEW, DRAWING), the SolidWorks drafting template dialog appears. After you choose a template and select OK, a SolidWorks drawing window is displayed and Cadra is started. Each time you create a new drawing, new SolidWorks windows are created (nothing changes in the Cadra window).

**Mode 2: The variable `ENABLE_SOLIDWORKS_DRAWINGS` is set to `FALSE`.**

CadraWorks behaves the same as it did with Cadra Version 10.3. When you create a new drawing in SolidWorks (open the FILE menu and select NEW, DRAWING), Cadra is started and a SolidWorks window is not opened.

Upon installation of Cadra Version 10.4, the default setting of the variable is FALSE. Therefore, the default behavior is the same as with Cadra Version 10.3—SolidWorks drafting is not available to the user.

## **Cadra.ini Variable for a DRT File External Reference Units Scale Factor**

A new variable in the cadra.ini file enables you to indicate a units scale factor in a DRT file external reference. Some Cadam files include a units conversion factor in the scale factor of an external figure reference and some do not. Cadra always applies the units conversion factor when instantiating a figure into a drawing. If the Cadam DRT file contains this conversion factor, the scaling of the figure will be incorrect. The following variable enables you to indicate if a units conversion factor is present in the external figure references of the Cadam DRT files being read into Cadra:

`ADRA_DRT_EXTERNAL_FIGURE_UNITS_SCALE`

A variable setting equal to 0 (the default setting) indicates there is no units conversion factor in external figure references. A variable setting equal to 1 indicates there is a units conversion factor in external figure references.

## **NEC PC 98 Keyboard: NFER Key Assignment**

Complete this procedure to assign the NFER key on the NEC PC 98xx series platform:

1. Modify the cadra.ini file to include the following entry:  
`key29=29 "NFER"`
2. Load NEC98.LLB as an alternate keyboard layout. NEC98.LLB is located in the LIB subdirectory of the Cadra install directory.
3. Use Autogeometry to assign a Cadra function to the key:  
Open the TOOLS menu.  
Select the AUTOGEOMETRY option.  
Select the LAYOUT OPTIONS tab.  
Assign a function to the key following the instructions in your *Autogeometry User Guide*.

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## Cadra Design Drafting

### Autosave

The Cadra Autosave feature automatically saves your current drawing at a user-specified time interval. In previous versions of Cadra, Autosave caused problems when invoked during commands (such as Insight) that had temporary entities in database groups. The Autosave backup feature has changed to correct this problem.

When it is time for an automatic backup and there are active groups, the backup will wait. Cadra will remember that it wants to perform an automatic backup and will continue trying to do so until groups are not active and the backup is successful.

Note that if you let Cadra sit idle for a long time with an active group, automatic backup will not occur. Consider the following examples:

#### Example 1:

If you add geometry and then temporarily stop your current work (such as to attend a meeting) without entering a new function, the backup will not occur while you are away because the ADD group is in use. The backup will occur as soon as you enter a new Cadra function.

#### Example 2:

As another example, you might leave the cursor over geometry with Insight on (so the geometry is highlighted) and leave to attend a meeting. The backup will not occur while you are away because the temporary highlight group is in use. The backup will occur as soon as you enter a new Cadra function.

#### Example 3:

If you pick the DELETE function, select geometry to delete, and leave to attend a meeting, the backup will not occur while you are away because the DELETE group is in use. The backup will occur as soon as you enter a new Cadra function.

#### Example 4:

If you select geometry in the GROUP function and leave to attend a meeting, the backup will not occur while you are away because the TEMP group is in use. The backup will occur when you cancel the TEMP group (which is often used like a permanent select group).

## Customizable Toolbars (Windows User Interface Only)

You can move various pushbuttons on the toolbar to suit your preferences. You can move a button by holding down the shift key, selecting the tool button, and then dragging it to the desired location within the same toolbar. This feature is available only for the following pushbuttons:

System Toolbar	Create, Open, Save, Cut, Copy, Paste, Plot, About
Function Toolbar	Point, Line, Circle, Spline, Offset, Text, Dimension, Delete, Move, Trim, Appearance, FlexDesign, Figure, Views, Info, Group
Viewing Tools	Undo, Redraw, Zoom In, Zoom Out, Toggle Insight

## New Colors for Workplanes

Three new system colors related to Cadra NC were added to both the Windows and Cadra user interfaces. The following definitions apply:

WCS    Work Coordinate System  
MCS    Machine Coordinate System  
wp     workplane

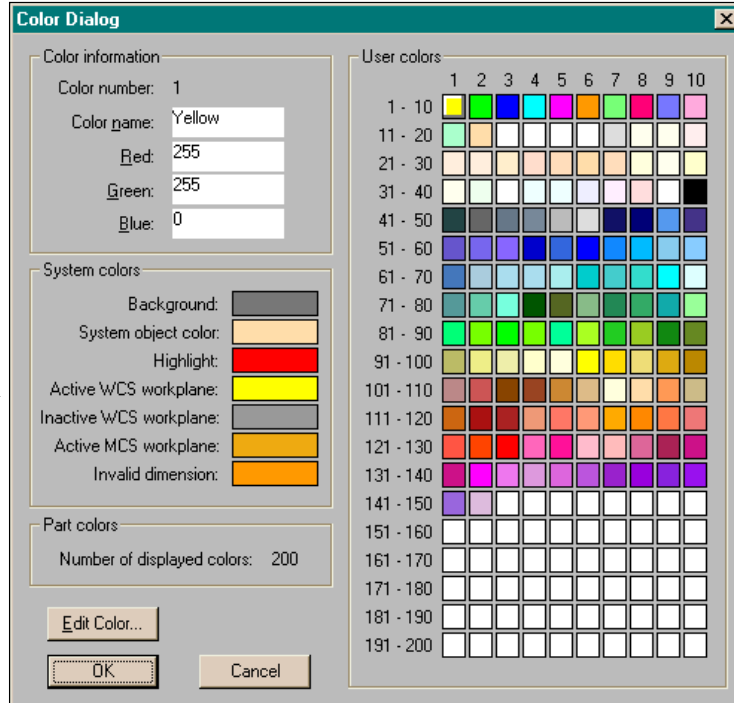
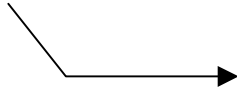
### **AA Windows User Interface**

The workplane colors listed in the System Colors group on the Color dialog box (accessed with the COLORS option on the SETUP menu) are:

Active WCS workplane  
Inactive WCS workplane  
Active MCS workplane

Refer to the dialog box shown on the next page.

New Workplane  
Color Options



### **AA Cadra User Interface**

The workplane colors listed on the menu when you pick PARAMETER, COLORS are:

- Act WCS wp
- Ina WCS wp
- Act MCS wp

## New Colors for Invalid Dimensions

A color option related to CadraWorks was added to both the Windows and Cadra user interfaces. It is the invalid dimension color used for dimensions which have become invalid as a result of changes in the CadraWorks part.

### **AA Windows User Interface**

The invalid dimension color is listed in the System Colors group on the Color dialog box (accessed with the COLORS option on the SETUP menu).

### **AA Cadra User Interface**

The invalid dimension color is listed on the menu when you pick PARAMETER, COLORS.

## Pen Scaling and Plot Inhibit Feature

You can indicate whether or not pens will scale with the plot if the plot scale is greater than 1.0. Generally, as you scale up (for example, pen 2 rather than 1), the thickness of the lines will increase (for example, double in size from pen 1 to pen 2). The new Cadra option for pen scaling enables you to indicate whether or not pens will scale with the plot.

In addition, you can control the plot scale of line fonts with the Inhibit Line Font Scale option. This includes three system line fonts: Dash, Centerline, and Phantom. It also includes user line fonts Userfont1 through Userfont18.

Currently, line font dash lengths scale up or down depending on the plot scale. If the plot scale is less than one, the dash lengths will decrease. If the plot scale is greater than one, the dash lengths will increase. The Inhibit option permits line fonts to remain at defined ON/OFF values independent of the plot scale.

For example, a dash line font is defined as ON for .125 inches and OFF for .0625 inches at a plot scale of 1. If the plot scale is scaled up to 2, this dash line font will be defined as:

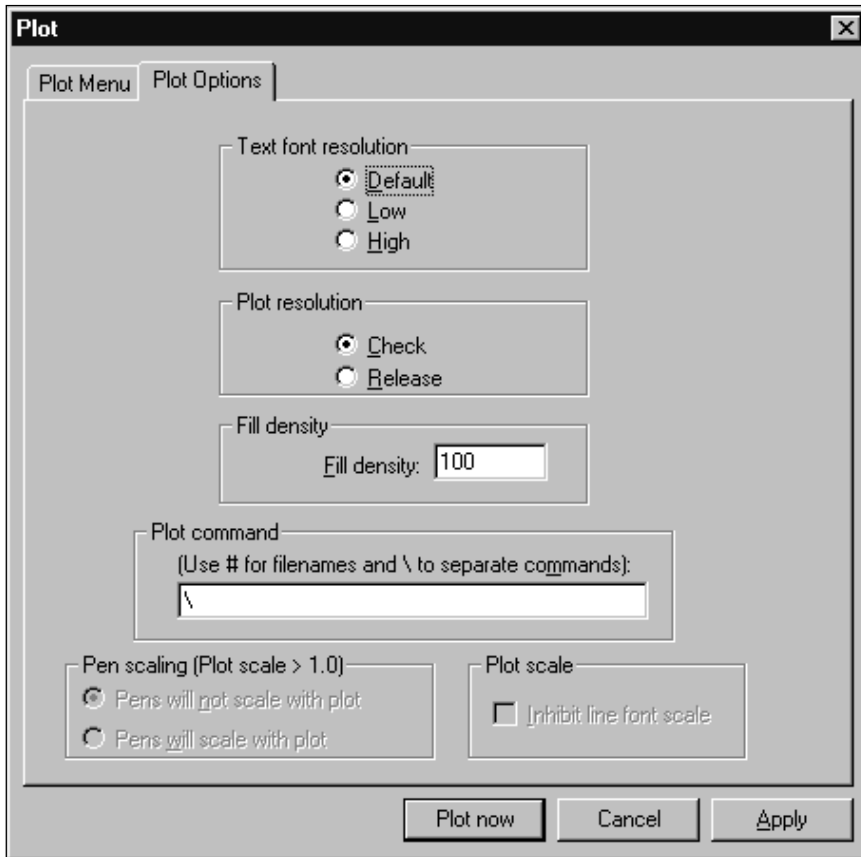
ON for .25 ( $.125 * 2$ ) and  
OFF for .125 ( $.0625 * 2$ )

If the plot scale is scaled down to .5, this dash line font will be defined as:

ON for .0625 (.125 / 2) and  
OFF for .03125 (.0625 / 2)

### AA Windows User Interface

The Pen Scaling options group was added to the Plot Options dialog box. Check either option, as desired. In addition, the Inhibit Line Font Scale option was added. It appears as a check box in the Plot Scale group. A check indicates “inhibit.” The default setting is unchecked and Cadra will scale line fonts.



Pen and Plot Scaling Options

## **AA Cadra User Interface**

Two new options were added to the PLOT menu:

- Plot scale
- Inhibit line font scale

The plot scale is inhibited when this option is highlighted.

For pen scaling, select one of the options to indicate whether or not pens will scale with the plot:

- Pen scaling:
  - Pens will not scale with plot
  - Pens will scale with plot

## **User-Specified Plotter Escape Sequences (Windows User Interface Only)**

Escape sequence options are available for the following plotter types:

- Restricted HPGL format
- Hewlett Packard 7475
- Hewlett Packard 7550
- Hewlett Packard 7585
- Hewlett Packard 7595

The procedure to set escape sequences for these plotters is:

1. Open the FILE menu.
2. Select PLOT.
3. On the Plot Menu dialog box, select one of the plotters listed above.
4. Click the Plotter Setup button. The Plotter Setup dialog box appears and contains a section entitled "Escape sequence" with two options. [Note that for the Hewlett Packard 7595, both options appear on a single line.]

Refer to the dialog box shown on the next page.



Escape Sequence Options

The “Default” option sets the escape sequence to the default which is the escape character, hexadecimal value 0x1b.

The “User specified” option enables you to type an escape sequence other than the default.

### Figure Color Display

The Figure Color Display feature provides flexibility in assigning colors to inserted figures. In this way, the figure color can be used to visually set figures apart from other types of entities in the drawing.

With SETUP, PREFERENCES (Windows User Interface) or PARAMETER (Cadra User Interface), you can select from three different figure color modes:

<p>Geometry Color Mode</p>	<p>(This is how “figure color” worked in Cadra prior to Version 10.2.) When this mode is active, each individual geometric entity which makes up the figure will display with its own color. The color with which a constituent entity is displayed is either:</p> <ul style="list-style-type: none"> <li>■ The color assigned to the layer on which the entity resides (see PARAMETER, LAYER COLOR). Note that this is the layer of the constituent entity, NOT the layer of the figure instance.</li> <li>■ The color assigned to the entity using the APPEARANCE, COLOR MENU. If a color is assigned to an entity using the APPEARANCE function then the layer color will be ignored.</li> </ul> <p>In Geometry Color Mode, assigning a color to the figure instance using the APPEARANCE function has no effect on the color used to display the figure.</p> <p>In Geometry Color Mode, it does not matter what color is assigned to the layer upon which the figure instance was inserted.</p>
<p>Instance Color Mode</p>	<p>When this mode is active, the entire figure is displayed in the same color, which is either:</p> <ul style="list-style-type: none"> <li>■ The color assigned to the layer on which the figure instance resides.</li> <li>■ The color assigned to the figure instance using the APPEARANCE function.</li> </ul> <p>In Instance Color Mode, it does not matter what layer the entities which comprise the figure individually reside upon.</p> <p>In Instance Color Mode, assigning a color to a figure's constituent entity with the APPEARANCE function has no effect on the color used to display that entity in the figure.</p>
<p>Default Appearance Color Mode</p>	<p>When this mode is selected, you are prompted to select a default appearance color.</p> <p>When this mode is active, any new figures subsequently inserted are automatically assigned the default appearance color (as though you had used APPEARANCE, COLOR MENU and selected that figure).</p>

Default Appearance Color Mode <i>(continued)</i>	This mode is advantageous if you deal with drawings having large numbers of figures and prefer to have groups of figures displayed with the same color.
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The Figure Color Display feature affects the color with which figures are plotted. You may wish to use Instance Color Mode or Default Appearance Color Mode during CAD file construction/editing, but then change to Geometry Color Mode before plotting their cad file.

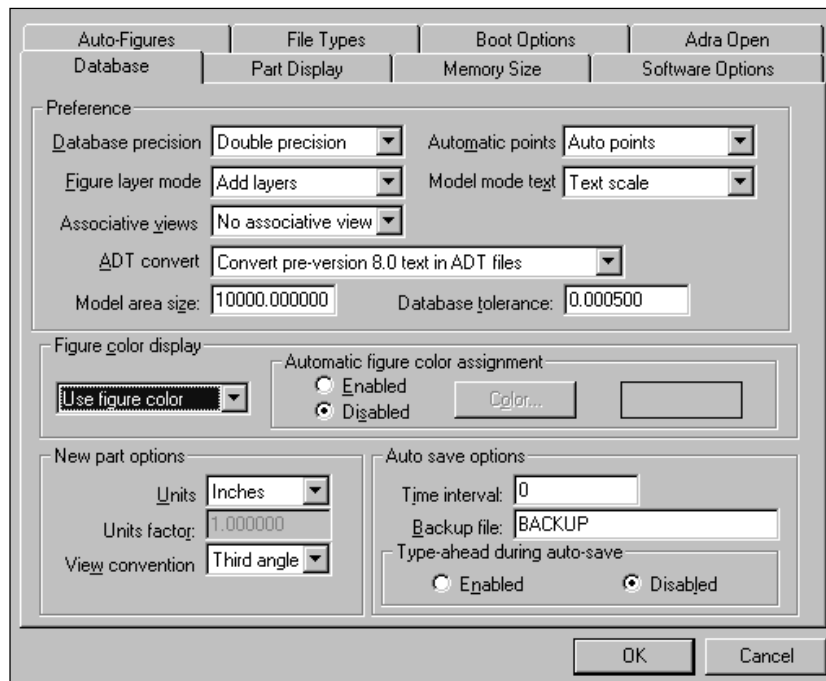
**This feature does not affect the translators.**

The figure color mode settings are saved with the Cadra part (.cad file) and also in the Cadra profile (prf.sys).

### AA Windows User Interface

Four new controls are added in the new Figure Color Display group on the Database tab (accessed from SETUP, PREFERENCES).

Figure Color  
Display Options



The controls are:

- A drop-down list offers two options: “Use entity colors” and “Use figure color.”
- Automatic figure color assignment can be enabled or disabled. Enable indicates that the figure color is automatically assigned when any subsequent new figures are inserted. This area is unavailable unless the default color is set to “Use figure color” mode.
- The Color... button invokes the Choose Color dialog. This button is unavailable if the color assignment is disabled.
- The “blank” button to the right of the Figure Color Display group is shaded with the active default appearance figure color (if applicable). This button is unavailable if the color assignment is disabled.

### **AA Cadra User Interface**

A new FIG COLOR option is be available through PARAMETER, SETUP. After picking FIG COLORS, options on the menu are:

- Geom color
- Inst color
- Dflt appear
- Return

When you select the DFLT APEAR option, the COLOR MENU is displayed and provides you with the opportunity to select the default appearance figure color. The previously selected default appearance figure color (if any) is highlighted in the menu.

## **Surface Display**

You can specify how surfaces are drawn by either:

- Using the number of display curves specified in each surface entity.  
This is the way surfaces are drawn currently. At the time the surface is created, information about drawing the surface is stored with the entity.

- Using the same number of display curves for all surface entities.

This is the new feature which enables you to override the number of display curves used to display surfaces. For “n” display curves in the U-direction and “m” display curves in the V-direction ( $n \geq 2$  and  $m \geq 2$ ), 2 boundary curves and  $n-2$  interior curves are drawn in the U-direction, and 2 boundary curves and  $m-2$  interior curves are drawn in the V-direction. The curves are equally spaced in U and V. Exceptions to the number of display curves are:

- Tabulated cylinders and ruled surfaces only display the boundary curves in the V-direction. No interior curves are drawn in the V-direction.
- Start and stop boundary curves may coincide if the surface is parameterized from 0 to  $2\pi$  (such as for a torus).

The surface entities affected are:

NURB spline surface  
UB spline surface  
quadric surface  
plane surface  
torus  
tabulated cylinder  
ruled surface

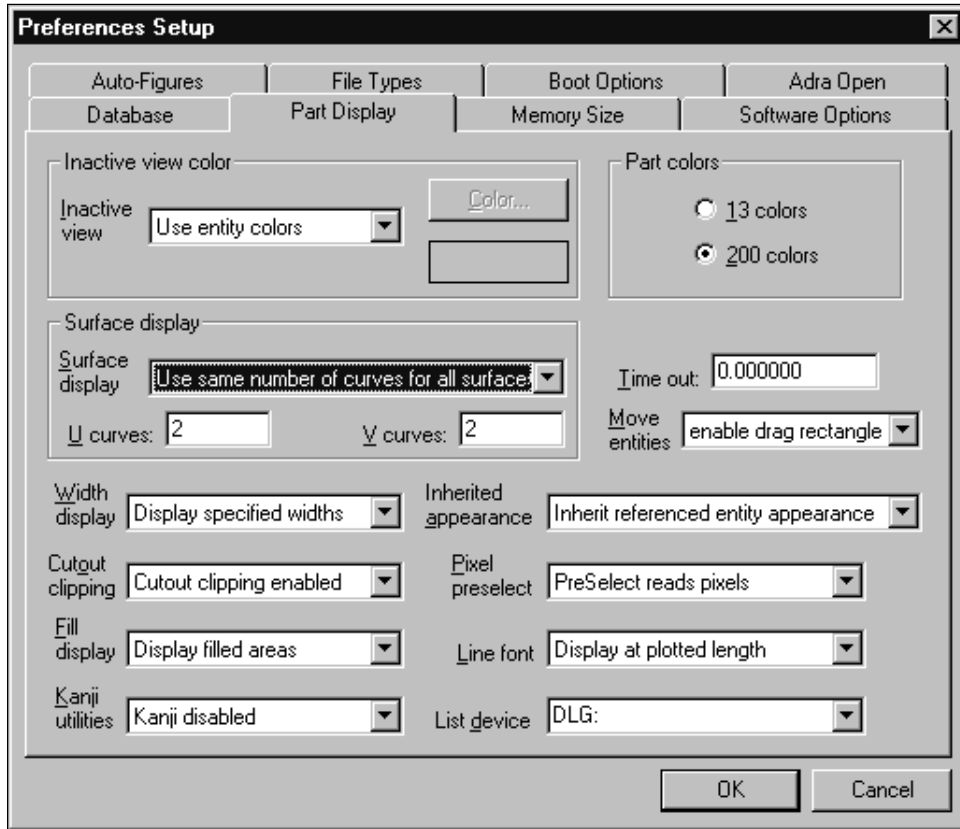
The surface display mode and the number of U and V display curves are stored in the Cadra profile. The entries are:

- `GRSrfDisplay=n` where n is:
  - 0 to use the number of display curves specified in each surface entity.
  - 1 to use the same number of display curves for all surface entities.
- `GRNumUCurves=u` where u is the number of display curves in the U-direction.
- `GRNumVCurves=v` where v is the number of display curves in the V-direction.

## AA Windows User Interface

Pick an option in the Surface Display group located on the Part Display tab of the Preferences Setup dialog (accessed from SETUP, PREFERENCES).

Surface  
Display  
Options



If you select “Use number of curves specified in entity,” the “U curves” and “V curves” edit control boxes are disabled. This is the way surfaces are currently drawn. At the time the surface is created, information about drawing the surface is stored with the entity.

“Use same number of curves for all surfaces” is the new option. Enter the U and V values in the “U curves” and “V curves” edit control boxes.

### **AA Cadra User Interface**

A new SRF DISPLY option is available through PARAMETER, SETUP. After picking SRF DISPLY, an acknowledgement message informs you that the current surface display mode will be issued. The message is either :

“Use number of curves specified in entity”

*Or:*

“Use same number of curves for all surfaces”

If you want to use the number of curves specified in the entity, pick the Choice box (USE ENTITY).

If you choose to use the same number of curves for all surfaces, key in the number of display curves in the U-direction and V-direction. The number of display curves in either direction must be greater than or equal to 2. You may key:

- Single number—the number of U-curves and the number of V-curves are both set to that number.
- Two numbers—the first number represents the number of U-curves and the second number represents the number of V-curves.

Or, press Return if you want to use the last keyed number of U and V display curves that are stored in the keystroke buffer.

The surface display mode and the number of U and V display curves are stored in the Cadra profile (PARAMETER, SETUP, SAVE).

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## **Cadra Design Drafting: Object Linking and Embedding (OLE) (Cadra for Windows Only)**

### **Object Linking and Embedding (OLE) Container**

Windows OLE container is supported with Cadra on the Windows NT, and Windows 95 platforms.

OLE is a Windows feature that combines data from multiple applications into one document. Container support refers to a document that contains references to other documents. For example, Cadra as a container refers to a Cadra document having references to Microsoft Word documents, AutoCAD documents, or any other document that supports OLE. A container document “contains” documents from other applications.

Cadra may have references to the document data of other applications. These references may be copied and inserted in Cadra as either a linked document or an embedded document. OLE retains the ability to modify the document data in the original application. Linked data will be updated in the original document and allowed to update automatically or manually in the Cadra document. Embedded data will be updated only in the Cadra document, not in the original document.

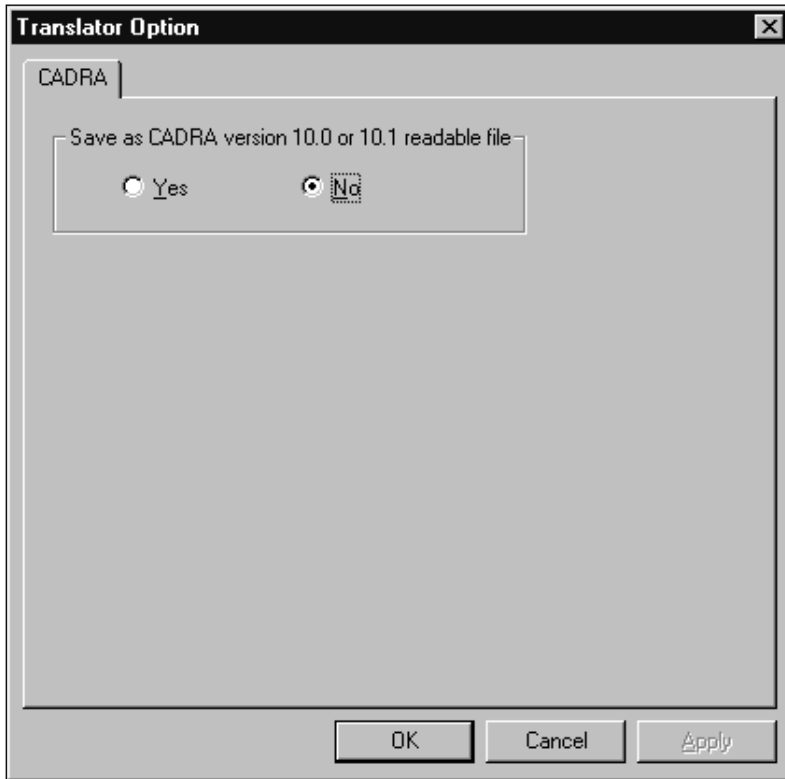
OLE objects are moved by moving the cursor over the object frame. A four-arrow cursor will appear. Drag this object by holding the left mouse button.

OLE objects are scaled or resized by moving the cursor over the object edge and selecting a grip. A double arrow cursor will appear. Drag a corner grip to scale the OLE object. Drag a middle grip to stretch the OLE object.

### **Cadra Database Support for OLE**

Cadra versions 10.2 and later support OLE. OLE objects are stored in the database as entities. Cadra versions 10.0 and 10.1 do not support OLE; therefore, when an unexpected OLE entity is encountered, the .cad file will not be readable.

When you select FILE, SAVE AS and click the Options button, the following dialog appears:



NO saves all OLE entities in the .cad file. This is the default setting.

YES removes all OLE entities before saving the .cad file.

Cadra databases are typically upward compatible but not downward compatible between major revisions of Cadra. Version 10.4 databases are readable by Cadra versions 10.0 and 10.1. The only exception to this rule at this time are the OLE entities. This feature will allow version 10.2 databases to be read by Cadra versions 10.0 and 10.1; but, the OLE entities will be removed.

## **Limitations**

Cadra documents cannot be embedded into Cadra documents. A linked object does not become an embedded object when the link is broken.

OLE objects will print only using the System printer. Cadra plot drivers will print the OLE object as an empty rectangle.

## **Object Names**

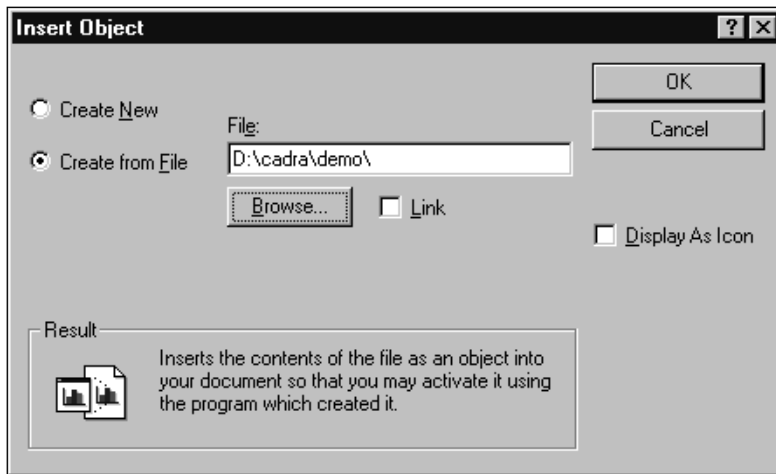
Depending on the application, examples of object names are:

- Linked Drawing Object — linked AutoCAD object
- Linked Document Object — linked Microsoft Word object
- Drawing Object — embedded AutoCAD object
- Document Object — embedded Microsoft Word object

## Linking Objects to a Cadra Drawing

You may want to insert a linked object into the Cadra drawing. This linked object may be edited within Cadra or externally by the original application. Changes to the linked object by an external application may be updated in Cadra. The linked object may be set to update automatically or manually.

1. Open the EDIT menu.
2. Select the INSERT NEW OBJECT option. The Insert Object dialog appears.



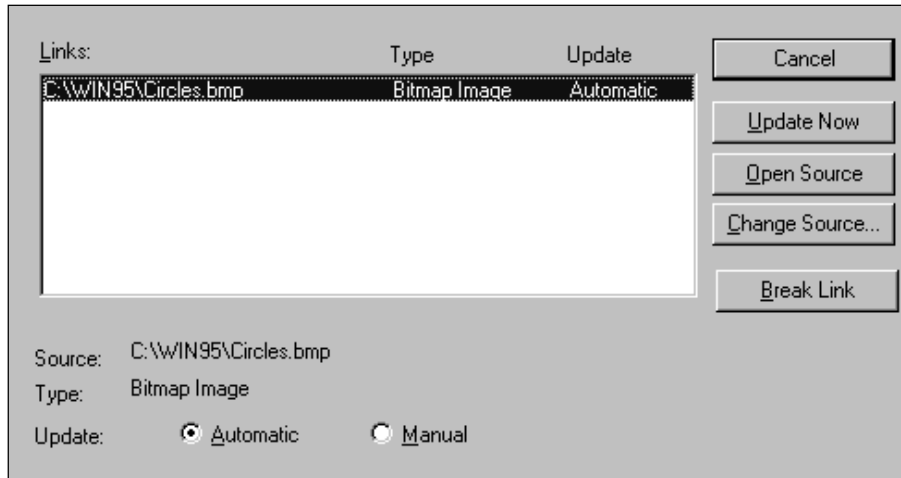
3. Select the CREATE FROM FILE option.
4. Key in or browse for a filename. The filename may be a Microsoft Word document, AutoCAD document, Excel document, etc.
5. Check the LINK box.
6. Click OK.

This OLE object is now linked to Cadra—Cadra is a container for the object. If this file is modified outside of Cadra, the changes are reflected in the Cadra document.

## Changing the Source File for a Link

You may want to change the link file of an existing link. For example, the original link filename may have changed or requests a different link file.

1. Open the EDIT menu.
2. Select the LINKS option. This dialog appears:



This dialog lists the linked file location, link type (CADRA for Cadra document, Drawing for AutoCAD document, Document for Microsoft document, etc.), and update type (Automatic or Manual). This dialog lists all current links in the Cadra document. It enables you to alter a selected link:

- Update the link automatically.
  - Update the link manually.
  - Change the source file for the link.
  - Break the link to disconnect the link from its link source.
  - Invoke source file for modification.
3. For this procedure (to change the source file for the link), select the link to be updated.
  4. Click the Change Source button. The Change Source dialog appears.
  5. Choose a new link file.

6. Click OK.
7. Click the Update Now button. The old OLE object is replaced by the new OLE object.

## **Breaking Links**

When you break a link, you disconnect the link from its source link. You may need to break a link when the linked data no longer needs to be updated. The object remains in Cadra but cannot be updated with the original application.

1. Open the EDIT menu.
2. Select the LINKS option. The dialog shown on the previous page appears.
3. Choose the OLE link to break by selecting the link filename.
4. Click the Break Link button. A Break Links confirmation dialog will appear.
5. Select YES to disconnect this link from its link source. The OLE object is no longer a linked object.

## **Creating an Embedded Object in Cadra**

1. Open the EDIT menu.
2. Select the INSERT NEW OBJECT option. The Insert Object dialog appears.
3. Select the CREATE NEW option.
4. Choose the object type to be inserted. (The object type list is determined by the system registry.)
5. Click OK to complete the selection.

The chosen application is either spawned as a totally separate application or the Cadra application transforms into the application GUI.

Separate applications are usually completed by selecting UPDATE and EXIT on the FILE menu. In-place applications are exited by clicking outside the object's frame. This returns control back to the Cadra user interface.

Depending on the object type being inserted, different behaviors will occur:

Paintbrush does not support in-place editing; therefore, the Paintbrush application will be spawned as a separate application. The Paintbrush application will modify the items with options on the FILE menu. The standalone Paintbrush application contains SAVE and EXIT options on the FILE menu. The SAVE option will be changed to UPDATE and the EXIT will be changed to EXIT & RETURN TO CADRA IN 'name.cad' when Paintbrush is spawned through INSERT OBJECT within Cadra. UPDATE will update the current Paintbrush picture into the Cadra application. EXIT & RETURN TO CADRA will close the Paintbrush application and return to Cadra.

Microsoft Word supports in-place editing; therefore, the Cadra application will appear to be a Microsoft Word application. Both corner and middle grips will surround the Microsoft Word document within Cadra. These corner and middle grips may be used to expand the Microsoft document. Clicking outside this gripped area will return control back to the Cadra application.

## **Embedding an Existing Object in Cadra**

With an embedded object, the content of the file is inserted as an object into the Cadra document. You can activate the object using the application which created it. Changes by another application to this document are not updated in Cadra, unlike linked objects.

1. Open the EDIT menu.
2. Select the INSERT NEW OBJECT option. The Insert Object dialog appears.
3. Select the CREATE FROM FILE option. The Insert Object dialog changes to a file entry dialog.
4. Browse to enter a filename. Do not select LINK for an embedded object. LINK is for linked objects only.
5. Click OK. The specified object is embedded into the Cadra document.

## Editing an Embedded Object in Cadra

There are two ways to edit an embedded object:

- Double-click in the embedded object. This will either spawn the application that originated the object or the Cadra application user interface will be changed for in-place editing.
- Select the embedded object (only one OLE object can be selected at a time).

To select the embedded object and edit it:

1. Open the FILE menu.
2. Select the 'doctype' OBJECT option. 'doctype' is the OLE object document type.
3. Select an option that is given OLE support by the application: EDIT, OPEN, etc.

## Submenus

The submenus may be different depending on the selected object type. The following are examples of submenus for various object types:

Application	Option	Description
Microsoft Word	Edit	Edit the Microsoft Word object in-place.
	Open	Edit the Word object in a separate Word application.
	Convert	Allow change to icon or convert to a Microsoft Word Picture versus a Microsoft Word Document.
Paintbrush	Edit	Edit the Paintbrush object in a separate Paintbrush application.
	Convert	Allow change to icon or convert to device independent bitmap.
Clipart	Replace	Replace the current clipart object.
	Convert	Allow change to an icon.

---

## Autogeometry

### PLANESURF Structure

A new Autogeometry structure named PLANESURF is added. It corresponds to the 3-D plane surface geometry in the Cadra database.

Note that PLANESURF is different from the existing Autogeometry structure PLANE3D. The PLANESURF structure is defined as follows:

Member	Type	Description
X	DOUBLE PRECISION	X coordinate of center
Y	DOUBLE PRECISION	Y coordinate of center
Z	DOUBLE PRECISION	Z coordinate of center
AX	DOUBLE PRECISION	X coordinate of axis in U direction
AY	DOUBLE PRECISION	Y coordinate of axis in U direction
AZ	DOUBLE PRECISION	Z coordinate of axis in U direction
BX	DOUBLE PRECISION	X coordinate of axis in V direction
BY	DOUBLE PRECISION	Y coordinate of axis in V direction
BZ	DOUBLE PRECISION	Z coordinate of axis in V direction
STARTU	DOUBLE PRECISION	start U parameter
STOPU	DOUBLE PRECISION	stop U parameter
STARTV	DOUBLE PRECISION	start V parameter
STOPV	DOUBLE PRECISION	stop V parameter
USPACE	DOUBLE PRECISION	spacing of display curves in U direction
VSPACE	DOUBLE PRECISION	spacing of display curves in V direction
FLAGS0	INTEGER	flags, not currently used
FLAGS1	INTEGER	flags, not currently used
FLAGS2	INTEGER	flags, not currently used

Member	Type	Description
FLAGS3	INTEGER	flags, not currently used
FLAGS4	INTEGER	flags, not currently used
FLAGS5	INTEGER	flags, not currently used
FLAGS6	INTEGER	flags, not currently used
FLAGS7	INTEGER	flags, not currently used

The U-axis and the V-axis must be orthogonal. The start and stop parameters are generally between 0 and 1. The flags fields are not currently used.

USPACE is defined to be the approximate number of display curves from U=0.0 to U=1.0. VSPACE is defined to be the approximate number of display curves from V=0.0 to V=1.0.

The actual number of display curves is:

$\text{INT}(\text{STOPU} - \text{STARTU}) * \text{USPACE} + 1$  in the U-direction

$\text{INT}(\text{STOPV} - \text{STARTV}) * \text{VSPACE} + 1$  in the V-direction

## New Functions

Three groups of Autogeometry functions were added for Version 10.4. The following functions work with 3-D plane surfaces:

- READPS\$ to read an existing plane surface (see page 33).
- MODPS\$ to modify an existing plane surface (see page 35).
- WRITPS\$ to write a new plane surface (see page 37).

Two functions were added to evaluate 2-D and 3-D NURB splines:

- EVALNS\$ to evaluate a point on a 2-D NURB spline (see page 39).
- EVALN3\$ to evaluate a 3-D point on a 3-D NURB spline (see page 41).

These functions read and modify entity names:

- READST\$ is an existing Autogeometry function that has been modified to return the name status of an entity whose name can be modified with a MODNM\$ (TYPE = 4) sub-routine call (see page 43).
- MODNM\$ is a new Autogeometry function that allow modification of entity names (see page 45).

## READPS\$

<b>Description</b>	Read a 3-D plane surface geometry from the database.		
	Function Call:	PLANE = READPS\$ (SEL)	
<b>Input</b>	Arguments:	SEL	SELECT structure for a 3-D plane surface.
	Implicit:	Nothing.	
	Restrictions:	None.	
<b>Output</b>	Function Return:	PLANE	PLANESURF structure for 3-D plane surface geometry.
	Errors:	An error will occur if the select structure is not for a 3-D plane surface.	

### Example

```
C EXAMPLE OF USE OF READPS$ AUTOGEOMETRY FUNCTION.
  PROGRAM READPS
C PROGRAM TO READ A 3D PLANE SURFACE AND DISPLAY ITS DATA
C
  SELECT SEL
  PLANESURF PLANE
C
C ***** START OF CODE *****
C
  SEL = GETSEL$(31, 0, 0, 'SELECT 3D PLANE SURFACE')
  PLANE = READPS$(SEL)
```

C

```
CALL PRINT$('CENTER = %h, %h, %h\n', PLANE.X, PLANE.Y, PLANE.Z)
CALL PRINT$('UAXIS = %h, %h, %h\n', PLANE.AX, PLANE.AY, PLANE.AZ)
CALL PRINT$('VAXIS = %h, %h, %h\n', PLANE.BX, PLANE.BY, PLANE.BZ)
CALL PRINT$('STARTU = %h, STOPU = %h\n', PLANE.STARTU,
1 PLANE.STOPU)
CALL PRINT$('STARTV = %h, STOPV = %h\n', PLANE.STARTV,
1 PLANE.STOPV)
CALL PRINT$('USPACE = %h, VSPACE = %h\n', PLANE.USPACE,
1 PLANE.VSPACE)
CALL PRINT$('FLAGS0,1,2,3 = %d, %d, %d, %d\n',
1 PLANE.FLAGS0, PLANE.FLAGS1, PLANE.FLAGS2, PLANE.FLAGS3)
CALL PRINT$('FLAGS4,5,6,7 = %d, %d, %d, %d\n',
1 PLANE.FLAGS4, PLANE.FLAGS5, PLANE.FLAGS6, PLANE.FLAGS7)
```

C

```
END
```

## MODPS\$

<b>Description</b>	Modify a 3-D plane surface geometry in the database.
	Subroutine Call: CALL MODPS\$ (SEL, PLANE)
<b>Input</b>	Arguments: SEL      SELECT structure for a 3-D plane surface. PLANE    PLANESURF structure to write.
	Implicit:            Nothing.
	Restrictions:      None.
<b>Output</b>	Effects:            The 3-D plane surface entity identified by the SELECT structure will be modified.
	Errors:             An error will occur if the select structure is not for a 3-D plane surface.

### Example

```
C EXAMPLE OF USE OF MODPS$ AUTOGEOMETRY SUBROUTINE.
  PROGRAM MODPS
C PROGRAM TO MODIFY A 3D PLANE SURFACE AND DISPLAY ITS DATA
C
  SELECT SEL
  PLANESURF PLANE
C
C ***** START OF CODE *****
C
  SEL = GETSEL$(31, 0, 0, 'SELECT 3D PLANE SURFACE')
  PLANE = READPS$(SEL)
C
  CALL PRINT$('ORIGINAL 3D PLANE SURFACE:\n')
  CALL PRINT$('CENTER = %h, %h, %h\n', PLANE.X, PLANE.Y, PLANE.Z)
  CALL PRINT$('UAXIS = %h, %h, %h\n', PLANE.AX, PLANE.AY, PLANE.AZ)
  CALL PRINT$('VAXIS = %h, %h, %h\n', PLANE.BX, PLANE.BY, PLANE.BZ)
  CALL PRINT$('STARTU = %h, STOPU = %h\n', PLANE.STARTU,
1 PLANE.STOPU)
```

```

CALL PRINT$('STARTV = %h, STOPV = %h\n', PLANE.STARTV,
1 PLANE.STOPV)
CALL PRINT$('USPACE = %h, VSPACE = %h\n', PLANE.USPACE,
1 PLANE.VSPACE)
CALL PRINT$('FLAGS0,1,2,3 = %d, %d, %d, %d\n',
1 PLANE.FLAGS0, PLANE.FLAGS1, PLANE.FLAGS2, PLANE.FLAGS3)
CALL PRINT$('FLAGS4,5,6,7 = %d, %d, %d, %d\n',
1 PLANE.FLAGS4, PLANE.FLAGS5, PLANE.FLAGS6, PLANE.FLAGS7)

```

C

```

PLANE.X = GETFLT$('ENTER X COORDINATE FOR PLANE CENTER')
PLANE.Y = GETFLT$('ENTER Y COORDINATE FOR PLANE CENTER')
PLANE.Z = GETFLT$('ENTER Z COORDINATE FOR PLANE CENTER')
PLANE.AX = GETFLT$('ENTER X COORDINATE FOR PLANE UAXIS')
PLANE.AY = GETFLT$('ENTER Y COORDINATE FOR PLANE UAXIS')
PLANE.AZ = GETFLT$('ENTER Z COORDINATE FOR PLANE UAXIS')
PLANE.BX = GETFLT$('ENTER X COORDINATE FOR PLANE VAXIS')
PLANE.BY = GETFLT$('ENTER Y COORDINATE FOR PLANE VAXIS')
PLANE.BZ = GETFLT$('ENTER Z COORDINATE FOR PLANE VAXIS')
PLANE.STARTU = GETFLT$('ENTER START U PARAMETER')
PLANE.STOPU = GETFLT$('ENTER STOP U PARAMETER')
PLANE.STARTV = GETFLT$('ENTER START V PARAMETER')
PLANE.STOPV = GETFLT$('ENTER STOP V PARAMETER')
PLANE.USPACE = GETFLT$('ENTER USPACE')
PLANE.VSPACE = GETFLT$('ENTER VSPACE')
CALL MODPS$(SEL, PLANE)
PLANE = READPS$(SEL)

```

C

```

CALL PRINT$('\nMODIFIED 3D PLANE SURFACE:\n')
CALL PRINT$('CENTER = %h, %h, %h\n', PLANE.X, PLANE.Y, PLANE.Z)
CALL PRINT$('UAXIS = %h, %h, %h\n', PLANE.AX, PLANE.AY, PLANE.AZ)
CALL PRINT$('VAXIS = %h, %h, %h\n', PLANE.BX, PLANE.BY, PLANE.BZ)
CALL PRINT$('STARTU = %h, STOPU = %h\n', PLANE.STARTU,
1 PLANE.STOPU)
CALL PRINT$('STARTV = %h, STOPV = %h\n', PLANE.STARTV,
1 PLANE.STOPV)
CALL PRINT$('USPACE = %h, VSPACE = %h\n', PLANE.USPACE,
1 PLANE.VSPACE)
CALL PRINT$('FLAGS0,1,2,3 = %d, %d, %d, %d\n',
1 PLANE.FLAGS0, PLANE.FLAGS1, PLANE.FLAGS2, PLANE.FLAGS3)
CALL PRINT$('FLAGS4,5,6,7 = %d, %d, %d, %d\n',
1 PLANE.FLAGS4, PLANE.FLAGS5, PLANE.FLAGS6, PLANE.FLAGS7)

```

C

```

END

```



```

PLANE.USPACE = GETFLT$('ENTER USPACE')
PLANE.VSPACE = GETFLT$('ENTER VSPACE')
PLANE.FLAGS0 = 0
PLANE.FLAGS1 = 0
PLANE.FLAGS2 = 0
PLANE.FLAGS3 = 0
PLANE.FLAGS4 = 0
PLANE.FLAGS5 = 0
PLANE.FLAGS6 = 0
PLANE.FLAGS7 = 0
SEL = WRITPS$(PLANE)
PLANE = READPS$(SEL)
C
CALL PRINT$('CENTER = %h, %h, %h\n', PLANE.X, PLANE.Y, PLANE.Z)
CALL PRINT$('UAXIS = %h, %h, %h\n', PLANE.AX, PLANE.AY, PLANE.AZ)
CALL PRINT$('VAXIS = %h, %h, %h\n', PLANE.BX, PLANE.BY, PLANE.BZ)
CALL PRINT$('STARTU = %h, STOPU = %h\n', PLANE.STARTU,
1 PLANE.STOPU)
CALL PRINT$('STARTV = %h, STOPV = %h\n', PLANE.STARTV,
1 PLANE.STOPV)
CALL PRINT$('USPACE = %h, VSPACE = %h\n', PLANE.USPACE,
1 PLANE.VSPACE)
CALL PRINT$('FLAGS0,1,2,3 = %d, %d, %d, %d\n',
1 PLANE.FLAGS0, PLANE.FLAGS1, PLANE.FLAGS2, PLANE.FLAGS3)
CALL PRINT$('FLAGS4,5,6,7 = %d, %d, %d, %d\n',
1 PLANE.FLAGS4, PLANE.FLAGS5, PLANE.FLAGS6, PLANE.FLAGS7)
C
END

```

## EVALNS\$

### Description

Evaluate a point on a 2-D NURB spline at a given parameter value.

Function Call: PT = EVALNS\$ (U, CURVE, SDATA)

### Input

Arguments: U (DOUBLE PRECISION) The parameter value at which to evaluate.

CURVE (SELECT or NURBBAY array)  
A SELECT structure for a 2-D NURB, or a NURBBAY array containing the 2-D NURB segment data.

SDATA (SPLINEDATA) Additional NURB spline data. Only required if CURVE is a NURBBAY array.

Implicit: Nothing.

Restrictions: The SELECT structure calling sequence is provided so that 2-D NURB spline evaluations can be performed when Autogeometry program memory is limited. In this case the 2-D NURB spline geometry doesn't need to be read into structures within the Autogeometry program. However, if program memory isn't limited, it is best to read the geometry prior to evaluating it, especially if multiple evaluations will be done.

### Output

Function Return: PT (POINT) The point on the 2-D NURB spline at the given parameter value.

### Example

```
C EXAMPLE OF USE OF EVALNS$ AUTOGEOMETRY FUNCTION.  
PROGRAM EVALNS  
C PROGRAM TO EVALUATE A POINT ON A 2D NURB SPLINE.  
C  
SPLINEDATA SDATA
```

```

      NURBBAY BAYS(100)
      POINT PT
      DOUBLE PRECISION U
      SELECT SEL, SELPT
      INTEGER CNT
C
C **** START OF CODE ****
C
C SELECT A 2D NURB.
C
      SEL = GETSEL$(25, 0, 0, 'SELECT 2D NURB SPLINE')
C
C READ NURB GEOMETRY.
C
      CNT = READNS$(SEL, 100, SDATA, BAYS)
      IF (CNT .EQ. SDATA.COUNT) GOTO 100
      CALL PRINTE$('NUMBER OF BAYS DO NOT MATCH')
      GOTO 999
C
C EVALUATE 2D NURB SPLINE AT U = 0.0, 0.25, 0.5, 0.75, 1.0
C USING GEOMETRY THAT WAS READ.  ADD POINTS TO DATABASE.
C
100  CONTINUE
      U = 0.0
          DO 200 I = 1, 5
              IF (I .EQ. 5) U = 1.0
              PT = EVALNS$(U, BAYS, SDATA)
              SELPT = WRITPT$(PT)
              U = U + 0.25
200  CONTINUE
C
C EVALUATE 2D NURB SPLINE AT U = 0.125, 0.375, 0.625, 0.875
C USING SELECT STRUCTURE.  ADD POINTS TO DATABASE.
C
      U = 0.125
          DO 300 I = 1, 4
              PT = EVALNS$(U, SEL, 0)
              SELPT = WRITPT$(PT)
              U = U + 0.25
300  CONTINUE
C
999  CONTINUE
      END

```

## EVALN3\$

**Description** Evaluate a 3-D point on a 3-D NURB spline at a given parameter value.

**Function Call:** PT3 = EVALN3\$ (U, CURVE, SDATA)

**Input**

**Arguments:**

- U** (DOUBLE PRECISION) The parameter value at which to evaluate.
- CURVE** (SELECT or NURBBAY3D array) A SELECT structure for a 3-D NURB, or a NURBBAY3D array containing the 3-D NURB segment data.
- SDATA** (SPLINEDATA) Additional 3-D NURB spline data. Only required if CURVE is a NURBBAY3D array.

**Implicit:** Nothing.

**Restrictions:** The SELECT structure calling sequence is provided so that 3-D NURB spline evaluations can be performed when Autogeometry program memory is limited. In this case the 3-D NURB spline geometry doesn't need to be read into structures within the Autogeometry program. However, if program memory isn't limited, it is best to read the geometry prior to evaluating it, especially if multiple evaluations will be done.

**Output**

**Function Return:** PT3 (POINT3D) The 3-D point on the 3-D NURB spline at the given parameter value.

### Example

```
C EXAMPLE OF USE OF EVALN3$ AUTOGEOMETRY FUNCTION.  
PROGRAM EVALN3  
C PROGRAM TO EVALUATE A POINT ON A 3D NURB SPLINE.  
C
```

```

SPLINEDATA SDATA
NURBBAY3D BAYS(100)
POINT3D PT
DOUBLE PRECISION U
SELECT SEL, SELPT
INTEGER CNT
C
C **** START OF CODE ****
C
C SELECT A 3D NURB.
C
    SEL = GETSEL$(25, 0, 0, 'SELECT 3D NURB SPLINE')
C
C READ NURB GEOMETRY.
C
    CNT = READN3$(SEL, 100, SDATA, BAYS)
    IF (CNT .EQ. SDATA.COUNT) GOTO 100
    CALL PRINTE$( 'NUMBER OF BAYS DO NOT MATCH')
    GOTO 999
C
C EVALUATE 3D NURB SPLINE AT U = 0.0, 0.25, 0.5, 0.75, 1.0
C USING GEOMETRY THAT WAS READ.  ADD POINTS TO DATABASE.
C
100 CONTINUE
    U = 0.0
        DO 200 I = 1, 5
            IF (I .EQ. 5) U = 1.0
            PT = EVALN3$(U, BAYS, SDATA)
            SELPT = WRITP3$(PT)
            U = U + 0.25
200 CONTINUE
C
C EVALUATE 3D NURB SPLINE AT U = 0.125, 0.375, 0.625, 0.875
C USING SELECT STRUCTURE.  ADD POINTS TO DATABASE.
C
    U = 0.125
        DO 300 I = 1, 4
            PT = EVALN3$(U, SEL, 0)
            SELPT = WRITP3$(PT)
            U = U + 0.25
300 CONTINUE
C
999 CONTINUE

```

END

## READST\$

<b>Description</b>	Read entity status.
	Function Call: STATUS = READST\$ (TYPE, SEL)
<b>Input</b>	Arguments: TYPE (INTEGER) Type of status to read. 1 thru 17—See existing documentation. 18—Read entity name status. SEL SELECT structure for the entity whose status is to be read.
	Implicit: Nothing.
	Restrictions: None.
<b>Output</b>	Function Return: STATUS (INTEGER) Entity status dependent upon TYPE. TYPE = 1 thru 17—See existing documentation. TYPE = 18—Entity name status. 0—Entity does not have a name that can be modified with a MODNMS\$ TYPE = 4 subroutine call. Note that view, detail, and axis entities have required names that cannot be edited arbitrarily. 0 is returned in this case. 1—Entity does have a name that can be modified with a MODNMS\$ TYPE = 4 subroutine call.

## Example

```
C EXAMPLE OF USE OF READST$ AUTOGEOMETRY FUNCTION.
  PROGRAM READST
C PROGRAM TO READ AN ENTITY'S NAME STATUS.
C
  SELECT SEL
  INTEGER STATUS
    INTEGER COUNT
    INTEGER NAME(130)
C
C **** START OF CODE ****
C
  SEL = GETSEL$(0, 0, 0, 'SELECT AN ENTITY')
  STATUS = READST$(18, SEL)
    IF (STATUS .EQ. 0) GOTO 100
  COUNT = READNM$(4, SEL, NAME)
  CALL PRINTA$('ENTITY NAME IS %s', NAME)
    GOTO 999
100 CONTINUE
  CALL PRINTA$('ENTITY DOES NOT HAVE A NAME')
C
999 continue
  END
```

## MODNMS

### Description

Modify the name of a view, figure, figure corresponding to a figure instance, or entity in the database.

Subroutine Call: CALL MODNMS (TYPE, SEL, COUNT, NAME)

### Input

Arguments:      TYPE      (INTEGER) Type of name to modify.

1—Modify name of current view. Will not be implemented now. Provided for compatibility with READNMS.

2—Modify name of current figure. Will not be implemented now. Provided for compatibility with READNMS.

3—Modify name of figure corresponding to current figure instance using SELECT structure. Will not be implemented now. Provided for compatibility with READNMS.

4—Modify name of current entity (such as a set) using SELECT structure.

SEL      SELECT structure for the entity whose name is to be modified. Only valid for TYPE = 3 or TYPE = 4.

COUNT      (INTEGER) The number of characters in the name, not including the terminating zero.

NAME      (INTEGER array) The ASCII characters of the name to write, terminated by a zero.

	Implicit:	Nothing.
	Restrictions:	Cannot modify the name of a view, detail, or axis entity using TYPE = 4. These entities have required names that cannot be edited arbitrarily.
<b>Output:</b>	Effects:	TYPE = 4 The name is written to the database. If no name previously existed, one is added. If COUNT = 0, then the name is deleted.
	Errors:	TYPE == 4 An error will occur if the entity is a view, detail, or axis entity.

### Example

```

C EXAMPLE OF USE OF MODNM$ AUTOGEOMETRY FUNCTION.
  PROGRAM MODNM
C PROGRAM TO MODIFY A NAME IN AN ENTITY.
C
  SELECT SEL
  INTEGER NAME(130)
  INTEGER COUNT
C
C ***** START OF CODE *****
C
  SEL = GETSEL$(0, 0, 0, 'SELECT ENTITY WITH A NAME TO MODIFY')
  COUNT = READNM$(4, SEL, NAME)
  CALL PRINT$('\nOLD COUNT = %d OLD NAME = %s\n', COUNT, NAME)
  COUNT = GETSTR$(NAME, 'ENTER NEW NAME')
  CALL MODNM$(4, SEL, COUNT, NAME)
  COUNT = READNM$(4, SEL, NAME)
  CALL PRINT$('NEW COUNT = %d NEW NAME = %s\n', COUNT, NAME)
C
  SEL = GETSEL$(0, 0, 0, 'SELECT ENTITY WITH A NAME TO DELETE')
  COUNT = READNM$(4, SEL, NAME)
  CALL PRINT$('\nOLD COUNT = %d OLD NAME = %s\n', COUNT, NAME)
  CALL MODNM$(4, SEL, 0, NAME)
  COUNT = READST$(18, SEL)
  IF (COUNT .EQ. 0)
1   CALL PRINT$('NAME HAS BEEN DELETED\n')
  IF (COUNT .EQ. 1)

```

```

1    CALL PRINT$('ERROR - NAME HAS NOT BEEN DELETED\n')
C
    SEL = GETSEL$(0, 0, 0, 'SELECT ENTITY WITHOUT A NAME')
    COUNT = READST$(18, SEL)
    IF (COUNT .EQ. 0) GOTO 100
    CALL PRINT$('\nERROR - ENTITY ALREADY HAS A NAME\n')
    GOTO 999
100  CONTINUE
    COUNT = GETSTR$(NAME, 'ENTER NEW NAME')
    CALL MODNM$(4, SEL, COUNT, NAME)
    COUNT = READNM$(4, SEL, NAME)
    CALL PRINT$('\nNEW COUNT = %d  NEW NAME = %s\n', COUNT, NAME)
C
999  continue
    END

```

---

## Translators: CDT and DXF

### Cadra Internal Figure Name Into Which a CDT Part is Read

The CREATE FIGURE option is available only for CDT files. No other translators are affected.

#### **AA Windows User Interface:**

When you open the FILE menu and select OPEN for CDT files, the Open dialog has a new CREATE FIGURE... button. Click this button to indicate a Cadra internal figure name into which the CDT part will be read.



#### **AA Cadra User Interface:**

The CREATE FIGURE option is available with the FILE function.

1. Pick the FILE function.
2. Pick the TRANSLATE option.
3. Pick the CDT MENU option.
4. Pick the CREATE FIG option.
5. Then, specify a Cadra internal figure name.

## DXF/DWG Entity Support

The DXF/DWG translator has had significant changes which include:

- Linetype names are correctly preserved during round-trip translations. Linetypes not recognized as being similar to standard Cadra linetypes are saved as user linetypes.
- Single-byte Katakana text translation code has been completely rewritten, resulting in faster and more accurate translation.
- Geometric symbols are preserved after round-trip translation. However, since Cadra supports more geometric symbols than AutoCAD, AutoCAD will not recognize some symbols.
- Color mapping formulas have been replaced with newer standardized methods, ensuring consistent color mapping of objects between Cadra and other CAD packages.

In addition, substantial read-side translation support for AutoCAD Release 14 entities has been added. The following table indicates whether or not a particular AutoCAD Release 14 entity type is supported during write or read DXF/DWG translations.

AutoCAD Release 14 Entity	Read Support	Write Supported
3DFACE	yes	no
3DSOLID	no	no
ARC	yes	yes
ATTDEF	yes	yes
ATTRIB	yes	yes
BODY	no	no
CIRCLE	yes	yes
DIMENSION	yes	yes
ELLIPSE	yes	no
HATCH	no	no
IMAGE	no	no
INSERT	yes	yes
LEADER	yes	no

AutoCAD Release 14 Entity	Read Support	Write Supported
LINE	yes	yes
LWPOLYLINE	no	no
MLINE	no	no
MTEXT	yes	no
OLEFRAME	no	no
OLE2FRAME	no	no
POINT	yes	yes
POLYLINE	yes	yes
RAY	yes	no
REGION	no	no
SEQEND	yes	yes
SHAPE	yes	no
SOLID	yes	yes
SPLINE	yes	no
TEXT	yes	yes
TOLERANCE	yes	no
TRACE	yes	no
VERTEX	yes	yes
VIEWPORT	somewhat	somewhat
XLINE	yes	no
ACAD_PROXY_ENTITY	no	no

## Viewports Limitations

Viewports are only partially supported. To ensure continuity of data between AutoCAD and Cadra, do not use viewports. Translate geometry from either paper space or model space.

## 3-D Support Limitations

Some geometry does not currently support 3-D data. Entity types that have been updated to support 3-D geometry effectively are:

- 3DFACE
- ARC
- ELLIPSE
- LEADER
- MTEXT
- POLYLINE
- RAY
- SOLID
- SPLINE
- TEXT
- TOLERANCE
- TRACE
- XLINE

At times, the Z-dimension is truncated from imports. The following entity types truncate Z-values of 3-D geometry. Translations may result in unexpected geometry requiring further clean-up.

- 3DLINE
- CIRCLE
- DIMENSION (due to LINE entity)
- LINE
- POLYLINE (making up 3d mesh surfaces, or faces)
- POINT

## **Additional Limitations**

Additional issues are involved with translating DXF or DWG files into Cadra:

- Cadra does not presently support an entity type such as SOLID. Mapping of this entity results in a polyline entity without any fill. In order to preserve data through translations, it would be best to use a polyline entity and add fill (rather than use SOLID).
- Polylines with “bulge” segments may produce unexpected results during translation.
- Circles with extrusion value will not translate as expected.

---

## Options: Cadra 3D and Raster

### Cadra 3D: Work Plane List

The WP LIST option under VIEWS, 3D MODEL, WP MENU displays a Work Plane List dialog (or submenu with the Cadra User Interface) if there are saved work planes. If there are no saved work planes, an error message appears: “No saved work planes.”

The Work Plane List dialog shows the saved work planes and enables you to select, delete, or name them.

### Raster: Pivot Point Definition

Previously when performing edit operations (scaling, mirroring, etc.) on a defined region of a raster image, the pivot point (or drag point) was always the center of the region. Now, you may define a pivot point about which the edit operations will occur.

When raster data is present in the current database, the PIVOT option appears under POINT, MOVE or OVLY MERGE.

1. Pick the POINT function.
2. Pick the MOVE or OVLY MERGE option.
3. Pick the PIVOT option.
4. Locate, select, or key in coordinates to define a pivot point.

Note that you can pick the Choice button, “Area midpt.” The pivot point will default to the center of the selected raster area that you are editing.

## **Raster: Rotation Angle Definition**

You can key in any angular value in degrees (rather than selecting from a fixed rotation angle of 90, 180, or 270 degrees). The new ROTATE option replaces the previous fixed rotation options.

1. Pick the POINT function.
2. Pick the MOVE or OVLVY MERGE option.
3. Pick the ROTATE option.
4. Key in a rotation angle.

## **Raster: Raster Resolution Control**

The new DPI option enables you to key in a resolution value to be used for rasterizing vector entities.

### **AA Windows User Interface:**

The DPI option is available with TOOLS, RASTERIZE.

1. Open the TOOLS menu.
2. Pick the RASTERIZE option.
3. Pick the DPI option.
4. Key in a resolution value in dots per inch.

### **AA Cadra User Interface:**

The DPI option is available with the FILE function.

1. Pick the FILE function.
2. Pick the TRANSLATE option.
3. Pick the RASTER MEN option.
4. Pick the DPI option.
5. Key in a resolution value in dots per inch.

---

## Cadra NC

### Cadra NC Defaults File: `prf_nc.sys`

You can now set default options for Cadra NC from a new file, `prf_nc.sys`, on DEC, HP, SUN O/S, SUN/SOLARIS, Windows NT, and Windows 95 systems. This defaults file is similar to the `prf.sys` file used to set options in Cadra. The `prf_nc.sys` file enables you to customize the work environment within Cadra NC.

Prior to Version 10.4, when you wanted to set a number of parameters, you set them each time you entered Cadra NC. Some parameters, such as color, are written to the APT file and may be set by a macro file. But many others such as those pertaining to tool display, are not written to the APT file and, therefore, had to be set manually each time.

Now, Cadra NC includes nearly all the parameters under the `PARAMETER` function and parameters previously set by environment variables (except for `CADRA_NC`). The environment variables are still supported but will be overridden by the settings in the `prf_nc.sys` file if the same parameter is referenced.

The system will use the first file called, `prf_nc.sys`, that appears in your `CADRA` path. Currently, there is no user interface support for the file—you must edit it manually using a system editor. The file is read in upon entry to Cadra NC. If you edit the file after entering Cadra NC, your changes will not take effect until you exit and return to Cadra NC.

You may use the menus to temporarily override most parameter settings during the current session. The parameters will be re-initialized to the settings in the `prf_nc.sys` when the file is re-executed or a new APT file is read.

Parameters supported for Cadra NC Version 10.2 and later are:

Identifier	Field	Description
<code>NCAutoLabel</code>	boolean	TRUE = generate labels.
<code>NCColorTool</code>	color	Color for tool images.
<code>NCColorMot</code>	color	Color for regular motion.
<code>NCColorProg</code>	color	Color for program motion.

Identifier	Field	Description
NCColorRapid	color	Color for rapid motion.
NCColorGeom	color	Color for regular geometry.
NCColorSurf	color	Color for surfaces, splines, and NURBs.
NCColorQuad	color	Color for quadrics.
NCColorPlane	color	Color for planes.
NCColorProf	color	Color for profiles and contours.
NCColorPat	color	Color for patterns.
NCColorVec	color	Color for vectors.
NCEditStep	integer	Step size in APT editor.
NCGeoLabel	boolean	TRUE = display NC labels.
NCGeoPlane	boolean	TRUE = display planes.
NCGeoSylStrt	double	Start distance from base of cylinder.
NCGeoCylStop	double	Stop distance from base of cylinder.
NCGeoConStrt	double	Start distance from base of cone.
NCGeoConStop	double	Stop distance from base of cone.
NCGeoConicSz	double	Conic display size.
NCGeoPlaneSz	double	Plane display size.
NCHelpDir	directory	Directory for help files.
NCMotCldata	boolean	TRUE = display TRACUT, INDEX, COPY.
NCMotTool	boolean	TRUE = display tool.
NCMotAxis	boolean	TRUE = display tool axis.
NCMotAll	boolean	TRUE = display tool at all points. FALSE = display tool at end points.
NCMotTlfig	boolean	TRUE = create auto tool figures.
NCPartDir	directory	Directory for APT files.
NCToolPosDis	boolean	TRUE = display tool position in error field.
NCErrLogDir	filename	Error log file.

Identifier	Field	Description
NCErrLogAppnd	boolean	TRUE = append to error log file.
GUI Only:		
NCErrDialog	boolean	TRUE = use dialog box for most errors.

The Cadra NC profile includes:

[CADRA\_NC Version ENT Tue\_14May96 PROFILE Version 1]

NCEditStep=5  
 NCGeoLabel=0  
 NCAutoLabel=1

NCToolPosDisp=0  
 NCErrDialog=1  
 NCErrLogDir="/home/logfile"  
 NCErrLogAppnd=0

NCGeoPlane=1  
 NCGeoCylStrt=0.0  
 NCGeoCylStop=5.0  
 NCGeoConStrt=0.0  
 NCGeoConStop=5.0  
 NCGeoPlaneSz=2.5  
 NCGeoConicSz=5.0

NCColorTool="Cyan"  
 NCColorMot="Green"  
 NCColorProg="Magenta"  
 NCColorRapid="Hot pink"  
 NCColorGeom="Yellow"  
 NCColorSurf="Magenta"  
 NCColorQuad="Yellow"  
 NCColorPlane="Blue"  
 NCColorProf="Pink"  
 NCColorPat="Orange"  
 NCColorVec="Aqua"

NCMotCldata=1

NCMotTlfig=1  
NCMotTool=1  
NCMotAxis=1  
NCMotAll=0

## Function Key Mappings

The mappings for several function keys have changed.

ALIGN MODEL replaces the obsolete CREATE SECT and will rotate the view to match with the active refsys.

HOR ISO replaces the obsolete ENTER SECT and will rotate the view to the horizontal isometric view.

INFO ERR replaces the obsolete EXIT SECT and will redisplay the latest error message on the screen.

## Spline and NURB Transfer Into Cadra NC

Splines and NURBs created in several CAD products can be transferred into Cadra NC through a series of conversions. Limitations of the previous version of Cadra NC are:

- NURBs are converted to 3<sup>rd</sup> degree splines with flat end conditions and no trim information.
- Splines are stripped of their end conditions and are brought in with no trim information.
- Entities that represent a logical unit are created as many tiny little entities.
- Splines (or tabcyls) and ruled surfaces built from splines and tabcyls are stored using a large amount of swap space.

## **NURB Machining Features**

- GENERATE NC brings in NURBs with end conditions and trim information preserved.
- The ability to machine NURBs through the motion commands GO and GODIR.
- GENERATE NC brings in splines with end conditions and trim information preserved.
- Improved memory management for tabcyls and splines (tabcyls are stored in the Cadra database as Bezier splines).
- Tabcyls are replaced by splines and/or NURBs in menus and messages.
- Older APT files containing the word tabcyl are still supported.
- Evaluator-based geometry creation using NURBs is provided.
- Create point at intersection of line and NURB.
- Create point at intersection of circle and NURB.
- Create circle tangent to a line and a NURB.
- Create circle tangent to a circle and a NURB.
- Create line perpendicular to a NURB through a point.
- Create line tangent to a NURB at a point.

## **Solution for the “Many Little Spline” Problem**

An interactive Cadra command, JOIN NURBS, was added for joining splines and NURBs that have the correct continuity criterion:

- They meet end-to-end.
- All lie in the same plane or are all true 3-D entities.
- They are tangent at the joined endpoints.

## Cadra NC Menus

The term “tabcyl” was replaced with “spline” and/or “NURB” in all menus and messages. NURBs use a new tag, NB1 - NB9999.

A feature was created to enable joining of splines and NURBs.

Cadra NC Version 10.2 or greater displays the following options when JOIN NURB is selected from the CADRA NURB menu:

SELECT GEO	<p>Enables you to select NURBs to join together. The location of the select of the first NURB is interpreted as a bias point to help determine the direction of traversal of the entity.</p> <p>If you pick the wrong entity, you may correct the entity selection by picking the Undo box. Only backing up by a single entity is allowed.</p>
CHAIN	<p>Saves you time and effort. The CHAIN option will continue from the entity selected until 1) it runs out of intersections that meet end-to-end, or 2) it finds multiple end-to-end intersections, or 3) it forms a closed geometry. To determine the direction of the chain when there are no entities selected, you should select the entity at the end nearest the start of the new NURB. If you pick the Undo box, all the entities highlighted in the chain are de-highlighted and the chain is aborted.</p> <p>Unlike the chain feature elsewhere in Cadra, the CHAIN option for JOIN NURB is 3-D and based on end-to-end matches only. CHAIN will stop as soon as an ambiguous direction or intersection is encountered. If chaining selects more entities than you want to include in the NURB, they can be removed from the chain by using the BACKUP option (described below).</p> <p>You can mix chaining and interactive entity selection freely. Generally chaining will stop when the system finds multiple intersections at the end of an entity—with one exception. If only one of the entities that has a matching endpoint lies in the same XY-plane as the preceding entity, the chain will continue on from that entity and the other entities with matching endpoints will be ignored.</p> <p>If the NURB is not yet complete, you may select another entity to start a new chain or pick SELECT GEO to continue selecting entities interactively.</p>

BACKUP	Enables you to remove entities from the end of the current selection. After you back up to the specified entity, the system returns to the previous selection mode (chaining or interactive entity selection).
NO REPLACE	Creates a new NURB without affecting the original geometry. This option is the default.
REPLACE	Removes the original NURBs and replaces them with a single NURB.
DONE	Indicates that you have completed data input. The system terminates the NURB combination by accepting the most recently selected entity into the contour, joins the NURBs, and deletes the original NURBs if the REPLACE option is selected.
ABORT	Ends the current operation.
RETURN	Contains an implied done and returns you to the GEOMETRY menu.

### Syntax for NURB Transfer

```
<snrb = NURB/CADRA, [<matrix, ]<scount, <degree, <flags, <startu, <stopu,
<nseg1, <nseg2, . . . . , <nsegn
```

where:

**<matrix** is "TRFORM, mat1"

**mat1** is a transformation matrix

**<scount** is the number of segments:  $m + K + 1 = N + 2$

**<degree** is the polynomial degree:  $m$

**<flags** is a bitmask indicating the following:

( DB\_BS\_W1 = 1 ) means all weights equal, ie. not rational.

( DB\_BS\_FIXE1TAN = 2 ) means start tangent specified.

( DB\_BS\_FIXE2TAN = 4 ) means end tangent specified.

( DB\_BS\_TANCL = 8 ) means closed with tangency.

( DB\_BS\_NATURAL = 16 ) means 2nd derivative 0 at end points.

**<startu** is the parameter value to start visibility (= 0)

**<stopu** is the parameter value to stop visibility (<= 1)

The geometric specification of a NURB segment **<segn** is:

```
<point, <uinc, <u, <weight
```

where:

**<point** is the control point for this segment

**<uinc** is the parameter increment for drawing at a window scale of 1  
**<u** is the parameter value or knot for this segment  
**<weight** is the weight for this segment

**Notes:**

The first "degree" number of segments and the last segment have no control points.  
The first "degree" number of segments and the last segment have no coefficients.  
The first "degree" number of segments and the last segment have no weights.  
The parameter value in the segment[degree] = 0.0.  
The parameter value in the segment[scount - degree - 1] = 1.0.  
The first used "uinc" value is the segment[degree].  
The last used "uinc" value is the segment[scount- degree-2].

**Syntax for Spline Transfer**

**<sspl = SPLINE/CADRA, [<matrix, ]<scount, <startbay, <stopbay, <startu, <stopu<sseg1, <sseg2, . . . ., <ssegn**

where:

**<matrix** is "TRFORM, mat1" where mat1 is a transformation matrix  
**<scount** is the number of segments  
**<startbay** is the starting bay number ( <= scount; usually 1)  
**<stopbay** is the ending bay number ( > 1; usually scount)  
**<startu** is the starting u parameter (usually 0)  
**<stopu** is the ending u parameter (usually 1)

The geometric specification of a spline segment <sspln is:

**<point1, <point2, <point3, <point4**

where:

**<point1** is the start point for this segment  
**<point2** is a control point for this segment  
**<point3** is a control point for this segment  
**<point4** is the end point for this segment

## Cadra NC Improved Error Handling

Informative error messages are displayed which are linked by a function key to more detailed information. In addition, all errors that occur in the current NC session are written to a log file. Improved error handling applies to the DEC, HP, SUN O/S, SUN/SOLARIS, Windows NT, and Windows 95 platforms.

Improved error handling has been implemented to provide:

- Useful and informative error messages.
- A text file shipped with the software containing all error numbers, error messages, and an optional detailed description.
- A log file that contains a list of all errors that occurred in the current NC session.
- A more complete (longer than one line) description for many error messages by pressing the appropriate function key on that platform keyboard:

F3 on UNIX platforms.

F4 on PC platforms.

### Error Message Output

The current version of Cadra NC outputs messages in the following format if a message is found:

```
<message , WARNING:      Line <line #   Status <stat #  
      ERROR:
```

Or, this format is used for an unknown error:

```
ERROR: <error #      Line <line #      Status <stat #  
WARNING:<warning #
```

### Notes:

User interface messages have no line number because no APT statement is created.

Some user interface messages bypass the error handling routine and contain only the message itself.

Status followed by the status number (which is an error code set by a low level utility) is optional and present only in certain error conditions occurring during processing of motion commands.

Cadra NC outputs all messages in the following format except for user interface errors which will be shown without the line number because an APT statement is not created:

```
WARNING: <error #, Line <line #:      <message  
ERROR:
```

For example:

```
ERROR: 908, Line 145: Illegal character in file name.
```

All error messages are listed in a text file that is on-line and sent to you as well as incorporated into the *Cadra NC User Guide*. The name of the file is “errors” and is located in the Cadra NC installation directory. It has the following format:

```
<Error #      <message      <description
```

where:

**<error #** is an error number unique throughout CADRA-NC

**<message** is the textual message

**<description** is additional information about the message

For example:

```
908  Illegal character in file name. The following characters are  
not permitted in an APT file name:
```

```
"$", ":", "-", "[", or "]".
```

All error messages are added to an error log file. The default name for this file is “ncerrlog” and it resides in the directory from which Cadra is executed. You may change the name of the log file via the prf\_nc.sys file by adding the following entry:

```
NCErrLogDir="/home/user/tool/nc/logfile"
```

This error log is initialized on entering Cadra NC unless the following entry appears in the prf\_nc.sys file:

```
NCErrLogAppnd=1
```

If you use the APPEND option, you must monitor it from time to time to ensure that it does not take up too much disk space.

The error number output is unique throughout all of Cadra NC. Different functional groups have been assigned various ranges of errors:

System errors:	1-999
Program	100-199
Execute	200-299
Edit	300-399
Info	400-499
Window	500-599
Geometry errors:	1000-2999
General:	1000-1099
Trimming	1000-1099
Options	1000-1099
2d Geometry:	1100-1999
Point	1100-1199
Pattern	1100-1199
Line	1200-1299
Circle	1300-1399
Ellipse	1400-1499
Hyperbola	1400-1499
Splines/NURB	1500-1599
Profile	1600-1699
Vector,	1700-1799
Matrix/Wkplane	1700-1799
3d Geometry:	2000-2999
Plane	2000-2099
Cylinder	2100-2199
Cone	2200-2299
Sphere	2300-2399
Torus	2400-2499
Ruled srf	2500-2599

Bsurf	2600-2699
Motion prg errors:	3000-3999
Motion errors:	4000-4999
General errors	4000-4099
Go/Godir/Go delta	4100-4199
Lathe	4200-4299
Cutpro	4300-4399
Pocket points	4400-4499
Pocket bsurf	4500-4599
Pocket island	4600-4699
Features errors:	5000-5999
Mach cntrl	5000-5099

## New Cadra NC Functions and Options

Several Cadra NC functions and options are either new or modified for Versions 10.4. They are:

Function	Option	Description
FEATURES		Help files are now supported for user-defined menus. The name of the help file must be added directly after the menu text it follows and be separated by a comma both before and after. The name of the file must end with the extension “.hlp” and the total length of the file name (including the extension) may not exceed 12 characters.
GEOMETRY	OPTIONS	The AUTO LABEL option has changed. Rather than requesting a response to the prompt, “Auto label (Y/N)?”, two options are available from which you should pick one:  AUTO LABEL is equivalent to a Y response to the previous prompt.  USER LABEL is equivalent to an N response.
INFO	ERROR	This option displays the most recent error on the screen.
MACH CNTRL		Help files are now supported for all machine control statements.
MACH CNTRL	MOT CTRL	A submenu for TAP cycles has been added to the CYCLE menu. It provides the same options as CYCLE, DRILL except that DRILL is replaced with TAP in the CYCLE.
MACH CNTRL	MOT CTRL	The THREAD options have changed as indicated after this table.
MACH CNTRL	COOLNT	The TAPKUL option has been replaced by the HIGH and THRU options.

Function	Option	Description
PARAMETER	COLOR	Three new options were added for workplanes, as described on page 11.
PARAMETER	CYLDR SIZE and CONE SIZE	The CY CO LEN option has been replaced by: CYLDR SIZE generates the statement “\$\$CYLNGT #,#” and applies only to cylinders. CONE SIZE generates the statement “\$\$COLNGT #,#” and applies to cones.
PARAMETER	MOTION	The CLDATA option has changed. Rather than requesting a response to the prompt, “Show CL motion (Y/N)?”, two options are available from which you should pick one: CLDATA Y is equivalent to a Y response to the previous prompt. CLDATA N is equivalent to an N response.
PROGRAM		The user interface for the ERR ON/OFF option was replaced with the following options with equivalent functionality: ERROR ON        \$\$ERRON Stops execution if an error is encountered. ERROR OFF        \$\$ERROFF Allow noncritical errors to pass through. DRV ERR ON        \$\$DRSERR 1 Stops execution if a drive error is encountered. DRV ERR OF        \$\$DRSERR 0 Issues a warning if a drive error is encountered.
WINDOW	EDIT ISO	Two new options have been added: ISO VER sets the isometric view to the standard isometric. ISO HOR sets the isometric view to the standard isometric 90 degrees about the X-axis.

## The THREAD Command

The THREAD command is used with both milling and turning machines. Refer to your post-processor documentation for the required post commands. Lathe threading is also supported in the Cadra NC standard FEATURES menu. Contact your Adra representative if your system currently does not have the standard FEATURES menu. THREAD provides the following options:

Option	Command	Description
Turn,TPI	THREAD/TURN,TPI,n	Specifies a thread lead along the machine Z in threads per inch.
Turn/LEAD	THREAD/TURN,LEAD,n	Specifies a thread lead along the machine Z.
Face/TPI	THREAD/FACE,TPI,n	Specifies a thread lead along the machine X in threads per inch.
Face/Lead	THREAD/FACE,LEAD,n	Specifies a thread lead along the machine X.
Mill/LEAD	THREAD/MILL,n	Specifies the length of the helical move the linear axis will take with the circle block (for Mill/drilling machines only).  Note: This command is modal and remains in effect for all subsequent circle records until cancelled by THREAD/MILL,OFF.
Mill/OFF	THREAD/MILL,OFF	Cancels helical interpolation.
Mill/ON	THREAD/MILL,ON	Reinstates previous THREAD/MILL,n command.

---

## CadraWorks

### Exploded View Procedure Modifications

The feature to generate an explode view of a solid assembly in CadraWorks has been modified. Chapter 6, pages 6-12 and 6-13, of the *CadraWorks User Guide* should be modified as follows. Steps 2, 3, and 11 have been replaced.

#### **AA Procedure for Creating, Saving, and Inserting an Exploded View**

This procedure enables you to rotate an exploded view to the desired orientation before bringing it into Cadra.

1. Open an assembly drawing in SolidWorks.
2. Click Insert, Exploded View.
3. In the Assembly Explode dialog box, click AutoExplode. The assembly explodes.  
Note that if a more customized explode configuration is desired, refer to “Exploding an Assembly View” in Chapter 5 of the *SolidWorks 97Plus user’s Guide*.
4. Rotate the exploded drawing to the desired orientation using the 3-D view rotation option.
5. From the View menu, select Orientation. The View Orientation dialog box appears.
6. Click the Add button and then key in a name for the view.
7. Click the Update button and then close the dialog box.
8. Pick the VIEWS function in Cadra Design Drafting.
9. Pick the SOLIDS option.
10. Pick the MODEL VIEW option.
11. Key in the name given to the previously saved view.  
*Or:*  
Choose LIST (in the Choice box) to display a list of selectable views, select one, and click OK.

**12.** Locate a point where you want the view to appear and press Enter.

The SolidWorks geometry will be projected to the designated location centered at the Cadra origin.

### **Cadra View of the Current SolidWorks View**

You can create a view in Cadra of the current SolidWorks view. The procedure to do so is the same outlined above (“Procedure for Creating, Saving, and Inserting an Exploded View”) with two exceptions: omit Steps 2 and 3.

---

## Notes