

# What's New in CADD5<sup>®</sup> 5i Revision 10.0

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CADD5 5i Revision 10.0

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# Preface

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*What's New in CADD5<sup>®</sup> 5i Revision 10.0* is an introduction to what has been added to CADD5 since CADD5 5 Revision 9.2.

## Related Documents

The following documents may be helpful as you use *What's New in CADD5 5i Revision 10.0*:

- *CADD5 5i Revision 10.0 Tips and Considerations*
- *CADD5 5i Revision 10.0 Read This First*

## Book Conventions

The following table illustrates and explains conventions used in writing about CADD5 applications.

Use of the Convention	Example	Explanation
Menu selections and options	List Section option, Specify Layer field, File menu	Indicates a selection you must make from a menu or property sheet or a text field that you must fill in.
User-selected graphic location	X, d <sub>1</sub> or P1	Marks a location or entity selection in graphic examples.
User input in CADD5 text fields and on any command line	<code>cvaec.hd.data.param</code> <code>tar -xvf /dev/rst0</code>	Enter the text in a CADD5 text field or on any command line.
System output	<code>Binary transfer complete.</code>	Indicates system responses in the CADD5 text window or on any command line.
Variable in user input	<code>tar -cvf /dev/rst0 filename</code>	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	<b>tagname</b>	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace <b>tagname</b> with an actual tag name.
CADD5 commands and modifiers	INSERT LINE TANTO	Shows CADD5 commands and modifiers as they appear in the command line interface.
Text string	"SRFGROUPA" or 'SRFGROUPA'	Shows text strings. You must enclose text strings with single or double quotation marks.
Integer	<i>n</i>	Supply an integer for the <i>n</i> .
Real number	<i>x</i>	Supply a real number for the <i>x</i> .
#	<code># mkdir /cdrom</code>	Indicates the root (superuser) prompt on command lines.
%	<code>% rlogin remote_system_name</code> <code>-l root</code>	Indicates the C shell prompt on command lines.
\$	<code>\$ rlogin remote_system_name</code> <code>-l root</code>	Indicates the Bourne shell prompt on command lines.



## Window Managers and the User Interface

According to the window manager that you use, the look and feel of the user interface in CADD5 can change. Refer to the following table:

**Look and Feel of User Interface Elements**

User Interface Element	Common Desktop Environment (CDE) on Solaris, HP, DEC, and IBM	Window Manager Other Than CDE on Solaris, HP, DEC, IBM, SGI, and NT
Option button	ON — Round, filled in the center OFF — Round, empty	ON — Diamond, filled OFF — Diamond, empty
Toggle key	ON — Square with a check mark OFF — Square, empty	ON — Square, filled OFF — Square, empty

## Online User Documentation

Online documentation for each book is provided in HTML if the documentation CD-ROM is installed. You can access the online collection in the following ways:

- From an HTML browser
- From the Information Access button on the CADD5 desktop or the LDM (Local Data Manager)

You can also access the online documentation directly from the CD-ROM without installing it.

From an HTML Browser:

1. Navigate to the directory where the documents are installed. For example,
  - `/usr/apl/cadds/data/html/htmldoc/` (UNIX)
  - `Drive:\usr\apl\cadds\data\html\htmldoc\` (Windows NT<sup>®</sup>)
2. Click the `mainmenu.html` file. A list of available CADD5 documentation appears.
3. Click the book title you want to view.

From the Information Access Button on the CADD5 Desktop or LDM:

1. Start CADD5.
2. Choose Information Access, the `i` button, in the top-left corner of the CADD5 desktop or the LDM.
3. Choose DOCUMENTATION. A list of available CADD5 documentation appears.
4. Click the book title you want to view.

From the Documentation CD-ROM:

1. Mount the documentation CD-ROM.

2. Point your browser to:

CDROM\_mount\_point/html/doc/mainmenu.html (UNIX)

CDROM\_Drive:\html\doc\mainmenu.html (Windows NT)

## Online Command Help

You can access the online command help directly from the CADD5 desktop in the following ways:

- From the Information Access button on the CADD5 desktop or the LDM
- From the command line

From the Information Access Button on the CADD5 Desktop or LDM:

1. Start CADD5.
2. Choose Information Access, the *i* button, in the top-left corner of the CADD5 desktop or the LDM.
3. Choose COMMAND HELP. The Command Help property sheet opens displaying a list of verb-noun combinations of commands.

From the Command Line: Type the exclamation mark (!) to display online documentation before typing the verb/noun combination as follows:

```
#01#!INSERT LINE
```

## Printing Documentation

A PostScript file is included on the CD-ROM for each document in the online book collection. See the first page of each online book for the document number referenced in the PostScript file name. The default documentation directories are:

- /usr/apl/cadds/data/html/htmlps/doc\_number.ps (UNIX)
- CDROM\_Drive:\usr\apl\cadds\data\html\htmlps\doc\_number.ps (Windows NT)

Printing from a UNIX System: Enter the following command at the system prompt:

```
lp -d printername filepath
```

Printing from Windows NT: To print from Windows NT to a network printer, at the MS-DOS prompt, use the `copy` command. The variable `filename` represents the file path and `systemname` represents the printer connection to the network.

```
copy filename \\systemname\printername
```

Check with your system administrator if you need more information.

## Resources and Services

For resources and services to help you with Parametric Technology Corporation (PTC) software products, see the *PTC Customer Service Guide*. It includes instructions for using the World Wide Web or fax transmissions for customer support.

## Documentation Comments

PTC welcomes your suggestions and comments. You can send feedback in the following ways:

- Send comments electronically to `doc_webhelp@ptc.com`.
- Fill out and mail the PTC Documentation Survey located in the *PTC Customer Service Guide*.



# What's New

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This document provides an overview of what is new in this CADD5 revision. These enhancements are grouped and described by product. For detailed information, please refer to the product-specific online document.

# New CADD5 Splash Screen and What's New Window

When you enter CADD5, an animated splash screen appears for a few seconds and disappears just before the CADD5 desktop is displayed. A What's New window displaying information on the new features in CADD5 appears along with the CADD5 desktop.

To exit the What's New window, click DONE or choose Quit from the Window Manager menu.

If you do not want to see the What's New window when you enter CADD5 again, clear the See This Next Time checkbox and click DONE. This displays a CADD5 warning window.

- If you do not want to display the What's New window again, click APPLY .
- To display the What's New window the next time you start CADD5, click CANCEL.

# Parametric Modeling

The following are enhancements in parametric modeling.

## History Validation and Repair Tools

You can do the following:

- Validate any command in the part history using the **Validate** option while in **REPLAY HISTORY** mode.
- Validate the commands in the part history using the **History** option of **CHECK DBASE**.
- Remove any defective command from the part history using **REMOVE INVALID\_CMD**.

## Measuring Distance

Use the **Maximum** option of **MEASURE LENGTH** to find the maximum distance between two selected entities.

## Single Modeling Environment

Use the **Associativity** option of **DEFINE CPLANE** to share a Cplane between both the **Explicit** and the **Parametric** environments. The Cplane is defined in the **Parametric** environment.

## Copying Geometry

You can use two new modifiers with the **Insert Ppart** command. **LAYEROFFSET** and **ULAYER** allow you to copy geometry from the source part to any layer in the target part.

## Replacing Sketches

With **REPLACE SKETCH**, you can replace an already applied sketch with another sketch. The replacement sketch can be one previously applied or it can be a sketch that has been saved, but not yet applied. When an applied sketch is used, the sketch is first copied under a new name.

## Using the Status Bar in Replay History Mode

You can use the status bar in `REPLAY HISTORY` mode to reflect the current replay status of the model and to use replay options. The status bar displays the index number with the verb and noun of the command that is being replayed. You can obtain the `REPLAY HISTORY` details by clicking `R` on the status bar.

**Figure 1-1 Status Bar in Replay History Mode**

Part : =USERS.RKADAM.PARTS.DISK	Drawing: A	Units: IN	R	C
---------------------------------	------------	-----------	---	---

The index of the current command appears in verb/noun format on the Replay status bar with the following options:

**Figure 1-2 Replay History Status Bar**

Replay Command 1: Insert Box	+	Index	End	Show	Back	C
------------------------------	---	-------	-----	------	------	---

- + — Increments history replay by 1.
- Index — Replays the part history up to the specified command index.
- End — Replays the model from its current displayed state to the end of the part history without exiting replay mode.
- Show — Shows the next `REPLAY HISTORY` stage.
- Back — Exits the Replay status bar.
- C — Closes the status bar.



# Interactive Surface Design (ISD)

The following are enhancements in ISD.

## Loft Surface

You can create a loft surface by blending a profile along one or more guide curves. The guides define an implicit geometric shape such as an arc with given radius. Methods available are line, arc, circle, and conic.

## Draft Angle Analysis

Draft angle shading allows you to analyze surface quality based on a given draft angle and “pull” direction.

## Dynamic Light Source

You can define different types of light sources for rendering a model. You can modify the light sources dynamically.

## Couple Curve

You can couple two generator curves to create a three-dimensional curve.

## Pton by Projection on a Surface

You can create a pton on a surface at the normal projection from the data point onto the surface.

## Pton Creation Method in Parametric History

The creation method of curve and surface ptons is saved in history if they are used by selected ISD commands.

## Data Set with Reference to an Entity

You can create data sets that reference entities such as parametric points, ends or midpoints of parametric curves, ISD curves, and edges of surfaces and shells.

## Consistent Handling for Data Points

You can display and manipulate the tangent and curvature constraints of a data point in a data set.

## Stretch Polycurve

You can stretch any segment of a polycurve by dragging a point lying on it.

## Trim, Join, Divide, and Blend for Polycurves

The `Derive Curve` commands, `Trim`, `Join`, `Divide`, and `Blend`, support polycurves.

## Measure Distances

You can measure distances between polycurves and shells.

# Hidden Line Removal (HLR)

The following are enhancements in HLR.

## Update Management

When you activate a drawing, you are informed whether the activated view needs to be updated. The view is highlighted in magenta, the default color. Two new menus are available:

- The Drawing Data menu provides information about the drawing and the views that need to be updated.
- The View Data menu provides information about the selected view.

## HIDE OBJECT

The ALL modifier has a new submodifier, CLIP. CLIP allows you to select only the entities within, and crossing, selected view boundaries. You can use CLIP with any of the other submodifiers, AEC, CURVE, SOLID, and SURFACE.

# Integrated 3D Sketcher

The following are enhancements in Sketcher.

## Creation of Equals Constraint

The Constraints menu has an additional option, Equals Constraint. With this option, you can constrain two lines of different length, so that the two lines are made equal in length. You can also constrain a pair of circles, or a circle and an arc with different radii, so that the pair of circles, or the circle and the arc, become equal.

## Auto Solve ON/OFF Modes

The Environment menu provides two modes of operation, `Auto Solve ON` and `Auto Solve OFF`, when you work with underconstrained sketch geometry.

`Auto Solve OFF` allows you to have more control of underconstrained geometry and prevent undesirable changes to a sketch when you add constraints.

The changed dimensions and the newly added constraints are buffered and added to the sketch when the model sketch is regenerated with the `Regenerate` option or until automatic solving is switched `ON`.

You can reset the value of the changed dimension or discard a buffered constraint with the `Reset` option.

## Creation of Fillet Dimension

You can create a radial dimension on a fillet and then export the fillet dimension to the Parametric environment after it has been fixed. Once exported to the Parametric environment, you can modify the dimension. The sketch is automatically regenerated. You can associate variables with these dimensions and use them in equations.

## Vpane Offset and Orientation

The Vpane is the virtual plane on which all sketch geometry is created and manipulated. After you have defined the Vpane, you can do the following:

- Offset the Vpane by specifying a distance for the offset.
- Change or reverse the direction of the offset.
- Rotate the Vpane by a specified angle.

## Selection of Entities in a Window

By defining a window border around several entities, you can select all of them.

The Selection by Window options are available from the Selection area on the top bar of CADD5, similar to the selection method in the Parametric environment.

## Zoom and Scroll Options

A common operation during a sketch session is to zoom or scroll a view or drawing. These options have been enhanced so that you can zoom and scroll a display even while you are creating an object. The various zoom and scroll options are available from the top bar of CADD5.

## Mirror Option

You can mirror selected sketch geometry in the same direction as the axis of symmetry.

The source and the mirrored geometry image are identical. The axis of symmetry and the mirrored geometry are associated with each other. There is also associativity between the mirrored objects and the source objects.

The mirror option is available from the Duplicate Entity option of the Environment globals area of CADD5.

# Concurrent Assembly Mock-Up (CAMU) and Parametric Multipart Design (PMD)

The following are enhancements in CAMU and PMD.

## Verify Object Command

With `VERIFY OBJECT`, you can obtain details regarding the mirroring plane and mirror status.

With `VERIFY OBJECT` in the Parametric environment, you can also retrieve information about the source part of a model associated with a component.

## Activate Model Command

When you activate a part that contains a part inserted in it with `INSERT PPART`, if the part image is not synchronized with its source part, a popup menu appears that asks if you want to update the part now.

## Disabling the CAMU Status Window

You can use a new environment variable, `USE_CAMU_STATUS_WINDOW`. By setting this variable to "no" in the `caddsrc` file, you may experience improved system performance while components are being locked or unlocked. In addition, the `CALLBACK_AD` and `CALLBACK_SERVER` processes are not started. `USE_CAMU_STATUS_WINDOW` is set to "yes" by default.

# Context-Sensitive Dimensioning (CSD)

The following are enhancements in CSD.

## Modifying Ordinate Dimensions and Leader Segments

You can modify ordinate dimensions and the corresponding leader segments. You can also modify leader segments by clicking on an end point and dragging.

## Modifying and Repositioning Label Text

You can modify label text. You can also drag and reposition label text.

## Modifying the Leader Line Position for RDIM and DDIM

You can modify leader line positions for RDIM and DDIM using the Modify Leader menu options.

## Modify the Text Height Ratio

You can modify the primary text height ratio using the Height Ratio option from the Dimension Properties Tolerance property sheet.

## Multiple Level Zooming

You can zoom back to the original or intermediate state of a feature, as required.

## Changing the Dimension Tolerance Precision

You can modify the primary and secondary dimension tolerance precision using the Precision option.

## Undeleting Entities

You can restore entities that have been deleted using the `DEL` key by pressing the `u` key. This applies only to the last delete operation, if no other operation has been performed after the deletion.

## Verifying Entities

You can obtain `VERIFY ENTITY` command output by selecting an entity and pressing the `v` key. The output is displayed in the CADD5 command window.

## Enabling the Getdata Origin Filter

You can select the origin of entities by pressing the `o` key after highlighting an entity. This enables Getdata to display temporary graphics on the origin of the entities. You can select the origin by clicking the temporary graphics.

## Modifying Extension Gaps

You can modify both extension gaps simultaneously by selecting the `Modify Both Gaps` option.



# Design and Drafting

The following are enhancements in Design and Drafting.

## Copy View

When you copy a view, any hidden line removal and sectioning done in the original view are also reflected in the copied view.

## Dimension Update Management

Dimensions are regenerated if a parent entity is changed. These regenerated dimensions can undergo any change. You can visually indicate this change by setting the Retain Dimensions option from the Dimension Setup menu to On, or by issuing the `SELECT DIMENSION RETAIN ON` command. A color code indicates the dimension change. You can disable the visual indication of changes by setting the Retain Dimension option to Off, or by issuing the `SELECT DIMENSION RETAIN OFF` command.

## Design Dimensions

The To 2-D in Selected Views option on the Design Dimension menu is the default. This option inserts design dimensions in a selected view as draw mode dimensions.

## Model Dimension Associativity

With solids, linear dimensions are associated with vertices rather than edges as the default. This results in more closely associated linear dimensions.

For wireframe entities, dimensions are associated with edges even if the vertex associativity is enabled.

## Select Cplane

You can select Cplanes in the Explicit environment, even if they are defined in the Parametric environment. However, you cannot modify them. The Cplanes defined in the Parametric environment must be made associative to be available in the Explicit environment. They are represented by a different naming convention than in the Explicit environment, as follows:

- All lowercase characters in the name are changed to uppercase.
- To differentiate shared Cplanes from the Explicit Cplanes, a suffix `-PARA-<plane-sequence-no>` is added to the Cplane name.

# CV-Hull

The following are enhancements in CV-Hull.

## Plate Drawings

You can generate a plate drawing with an automatically extracted cutting and shrinkage allowance if the plate has a cutting and shrinkage allowance defined for it.

## Stiffener Drawings

You can generate a stiffener drawing with a mitred joint on the flange or the web.

## Ship Drawings

You can generate a longitudinal section drawing of a ship by specifying the following:

- Drawing location
- View direction
- Longitudinal extent
- Vertical extent

## Traverse Section Drawings

You can generate traverse section drawings with a specified vertical extent; that is, the distance from the baseline.

## Tnode Properties for Stiffener Form Parts

The following Tnode properties are available for stiffener form parts:

- 18 = WELDTYPE property if a stiffener has fillet welds associated with it
- 19 = BEVELTYPE property if a stiffener has fillet welds associated with it
- 200 = Serial number in the Cutout Text Table
- 201 = Section type of a cutout in the Cutout Text Table
- 202 = Angle of a cutout in the Cutout Text Table

## Tnode Properties for Panel Form Parts

The following Tnode properties are available for panel form parts:

- 25 = Collar name
- 26 = Bracket name
- 27 = Panel plate serial
- 28 = Associated stiffener serial
- 29 = Associated plate serial

## Data Associated with Tnodes

Data associated with Tnodes is interpreted as follows:

- DWGOBJATTR - (integer value 99) Indicates that the Tnode belongs to the inserted symbol.
- DWGOBJIDX - (integer) Indicates the Tnode index base on which the symbol data is to be inserted. Valid value is:
  - 1 = Type sequence number

## nestparams File

The following parameters have been added to the `nestparams` file:

```
SHOW-OBJ-ID 0  
PANEL-PLATE-PREFIX P-  
PANEL-ASSOC-PLATE-PREFIX AP-  
PANEL-ASSOC-STIFF-PREFIX AS-
```

## Electrical Cabling

The following are enhancements in Electrical Cabling.

- You can check cableway segments for a fill percentage value above or below the specified minimum or maximum values, respectively.
- You can check a selected cableway segment and compare its segregation classes with the segregation classes of all the cables that pass through it for any incompatibility.

# Piping

The following are enhancements in Piping.

## GENERATE SPOOL

You can spool pipelines by defining a maximum spool length and a maximum number of bends by defining the modifiers `SPOOLMAX` and `BENDMAX`. When a pipeline is broken at a calculated cutting location by defining the maximum spool length or the maximum number of bends, the connect nodes inserted on the pipeline are removed during reconstruction.

The modifiers `SPOOLMAX` and `BENDMAX` are mutually exclusive with the `RECONSTRUCT` modifier.

## SELECT PPARAMETERS

You can specify the maximum number of bends and the maximum spool length for spooling a pipeline by using the modifiers `SPOOLMAX` and `BENDMAX`. `GENERATE SPOOL` uses these modifiers with the `CHECK` flag for spooling the pipeline. Spooling is possible using these two modifiers only if the `CHECK` flag is active.

## GENERATE PIPEDBASE

You can extract a database related to Pipe Bend and Pipe Endtype information from a pipeline model by using this command. The following two new tables have been introduced for this purpose:

- Pipeline Bend Information Table  
Pipe bends information is extracted to this table. A pipeline should have at least one bend element for this purpose.
- Pipeline Item Number Information Table  
Pipe item number information is extracted to this table. A pipe spool must be annotated by the `SAVE ANNOTATION` command and each element of the pipe spool must have the `ITEMNUMB` property.

# Advanced Structural Modeling

The following are enhancements in Advanced Structural Modeling.

## Task Sets

Most of the options on the task sets have been restructured so that they are logically based on their functionality. A new Profile task set has been added and all the options for creating standard profiles are grouped under it.

## Manipulating Structural Objects

You can manipulate the following types of structural objects:

- Axial type SObjects: structural elements, endcuts, clearance cutouts
- Membrane type SObjects: structural plates, flanges, brackets, collars
- Generic type SObjects: structural welds

## Creating Structural Welds

You can connect a single primary SObject to multiple secondary SObjects with a single STweld. This way, welding can be achieved by a single run of a welding machine.

## Editing Structural Objects

All the property sheets used for editing SObjects are similar to the property sheets used for manipulating SObjects.

You can add a cutting or shrinkage allowance for a plate or a stiffener. This specified cutting and shrinkage allowance can be edited later, if required.

## Generating a Structural Database

You can store information about a generic type of structural object and nonassociated types of welds from a model and store mass properties or attributes or both in an external RDBMS database.

## Selecting Structural Plates from a Stock File

A stock file gives information about the available stock of plates. In this file, plates are classified based on their thickness. When creating a structural plate, you can specify the value of the plate thickness manually or by using an existing stock file.

## Creating Structural Transition Elements

You can connect structural elements of the same section type by using a transition element. While the elements to be connected should be of the same section type, the sections can be of different sizes.

## Creating Structural Elements on a Structural Plate

You can create a structural element associated with a set of structural plates. A stiffener can be created successfully on plates with a different thickness or orientation only if the surface of the plates on which the stiffener is connected forms a single smooth surface.

## Creating Welded Flanges

You can create a welded flange by welding an STElement to a plate along a selected edge. A welded flange can be inserted only on plates with reference geometry or on plates with cutouts as coamings. The welded flange acts as a connector for the selected plates. You can insert a structural flange within or outside the plate boundary.

## Creating Structural Endcuts

You can create an endcut by cutting only the web of an STElement without affecting the flange. A new section type 203 (Web only endcut) has been added in the section library.

The end of a selected STElement can be recognized automatically by using the Auto option for creating an endcut. Also, the system automatically recognizes whether you have selected a web or a flange of an STElement for endcut creation.

## Creating Structural Joints

You can connect two linear structural elements (STElements) with a mitre type of joint. The selected elements are extended or shortened as necessary to make the proper joint. You can also remove the mitre joint between the structural elements.



# CADD5 Composites

The following are enhancements in CADD5 Composites.

## Generating IGES

A new option to generate an IGES file for design laminates is available through Composites.

## User Interface Changes

Changes and internal enhancements have been made to the property sheet of Flattening Layup Surface.

## Translators

AP203 —The modifier `use_direct_solid` has been added to the CADD5 parameter table.

## CVact

If a CADDs (Cue/Composer or CVact) menu is tacked, it continues to remain tacked even when you click Reset on the menu.

# CV-DORS

The following are enhancements in CV-DORS and CV-DORS CAMU.

## CV-DORS Interfaces

New interfaces allow you to:

- Build an Nspline by joining several curves at a specified continuity level and tolerance.
- Approximate a base Nspline object using the knot vector and parameterization of a reference Nspline object.
- Remove the upper limit of 3000 polygon points on surfaces to be offset and return the pointer to the new offset surface.

## CV-DORS CAMU Interfaces

New interfaces allow you to:

- Return the relative path name of the CADDs model part associated with the current component instance.
- Return the relative path name of the CADDs model part associated with the current class.