

# What's New in CADD5<sup>®</sup> 5 Revision 9.0

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CADD5 5 Revision 9.0

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# Preface

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*What's New in CADD5<sup>®</sup> 5 Revision 9.0* is an introduction to what has been added to CADD5 5 since CADD5 5 Revision 8.1.1.

## Concerns and Documentation Comments

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

In regard to documentation, PTC welcomes your suggestions and comments. You can send feedback in the following ways:

- Send comments electronically to the following address:

`doc_webhelp@ptc.com`

- Fill out and mail the PTC Documentation Survey in the service guide.

## Online User Documentation

Online documentation is provided in HTML. PostScript files of each book in the online collection are also provided. You can print the PostScript files for hardcopy books. The documentation media contains the PostScript files.

## Related Documents

The following documents may be helpful as you use *What's New in CADD5 5 Revision 9.0*:

- *CADD5 5 Revision 9.0 Tips and Considerations*
- *CADD5 5 Revision 9.0 Release Notes*
- *What's New in CADD5 5 Revision 8.1*

## Book Conventions

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

<b>Use of the Convention</b>	<b>Examples</b>	<b>Explanation</b>
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	cvaec.hd.data.param tar -xvf /dev/rst0	Enter the text in a CADD5 text field or on any command line.
System output	Binary transfer complete.	Indicates system responses in the CADD5 text window or on any command line.
Variable in user input	tar -cvf /dev/rst0 filename	Replace the variable with an appropriate substitute; for example, replace filename with an actual file name.
Variable in text	tagname	Indicates a variable that requires an appropriate substitute when used in a real operation; for example, replace tagname 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 string with single or double quotation marks.
Integer	n	Supply an integer for the <i>n</i> .
Real number	x	Supply a real number for the <i>x</i> .
#	# mkdir /cdrom	Indicates the root (superuser) prompt on command lines.
%	% rlogin remote_system_name -l root	Indicates the C shell prompt on command lines.
\$	\$ rlogin remote_system_name -l root	Indicates the Bourne shell prompt on command lines.



# What's New in CADD5 5 Revision 9.0

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This document provides an overview of what is new in CADD5 5 Revision 9.0. Several aspects of CADD5 have been enhanced. These enhancements and considerations are grouped and described by product. For detailed information, please refer to the product-specific online document.

## Support for HP-UX OpenGL Graphics

CADD5 5 Revision 9.0 includes support for the HP-UX OpenGL graphics driver on workstations configured with the Visualize-fx graphics card.

OPENgl is now available on Solaris.

# Advanced Structural Modeling (ASM)

## Bracket Types

New bracket types have been added, as follows:

- D
- E
- F
- K
- L

## DIVIDE STOBJECT

The `DIVIDE STOBJECT` command now supports reference geometry (for `STelement` and `STplate` only). In addition, it now contains a new modifier, `NOUNDIVIDE`.

You can divide the selected structural elements into segments.

## EDIT STOBJECT

The `EDIT STOBJECT` command has the following new modifiers:

- `CENTERLINE`
- `FILLET`
- `FLIP`
- `FLOCATION`
- `FOFFSET`
- `ROTATE`
- `SECOFF`

## EDIT STSCALAR

`EDIT STSCALAR` is a new command that allows you to edit entries stored in the part index table. You can rename viewed part entries, keeping viewed part—entity relations intact.

## Flanges

- You can now add the bend radius option during insertion, which allows a bend during unfolding.
- The `Endcut` function now works on folded flanges.

## GENERATE STOBJECT

You can automatically generate structural objects from parametric entities having parametric attributes. You can create the property library by saving all the basic parametric entities into property classes. Even if the basic geometric entities have all the attribute information, the selective generation of axial, membrane, or generic type of structural objects is possible with the help of this command.

## INSERT STCUTOUT

`INSERT STCUTOUT` section now includes justification and a fillet on/off option. The cutout is editable in the same manner as any `STElement` section.

Two new modifiers, `CENTERLINE` and `FILLET`, have been added to this command.

## INSERT STFLANGE

The `INSERT STFLANGE` command now includes the `WELDED` modifier and its submodifiers.

## INSERT STPLATE

You can now create a structural plate whose shape is defined in the section library using a new modifier, `SECTION`, of this command.

## Interacting Endcut

An interacting `Endcut` can now follow the curved contour of an interacting object.

## Material Indexing

Material indexing can be done with or without trailing zeros as shown below:

```
>0000  
>00001  
>00000002  
>0003  
>14
```

## MODIFY STPLATE

The `MODIFY STPLATE` command now allows selective replacement of definition entities and can extend them to divide an `stobject` (for `STplates` only) to support reference geometry.

## TAG and NAME Modifiers

The `TAG` and `NAME` modifiers now accept a list of text values while creating multiple `STobjects` at one time and appropriately assign the values to the created objects. The following commands include this enhancement:

- `CREATE STOBJECT`
- `DIVIDE STOBJECT`
- `EDIT STOBJECT`
- `EDIT STWELD`
- `INSERT STBRACKET`
- `INSERT STCOLLOR`
- `INSERT STCUTOUT`
- `INSERT STELEMENT`
- `INSERT STENDCUT`
- `INSERT STFLANGE`
- `INSERT STPLATE`
- `INSERT STSYMMETRIC`
- `INSERT STWELD`

# CADD5 Composites

## Design

- In the Display Stacking Sequence icon, an option for writing the thickness law into file has been added. (format is similar to read thickness law).
- In the Generate Sublaminates icon, an option has been added to generate the sublaminates/laminates. If laminate is selected, it will create LAMINATE without asking for the name.
- In Sample Point creation, you can display the sample point data for the current Laminate. At each sample point, number of plies for each orientation is displayed.
- In Create Ply Surface, Create Multipatch Surface Through Automatic Route is made active. Patch boundaries are calculated internally.
- 5. In Generate Manufacturing Ply, an option is available to create manufacturing trim surfaces. This will minimize the time in
  - Displaying mfg ply information
  - Mapping allocation lines on surface (with offset) in flat allocation
  - Surface allocation trim.
  - Draping trim.

This will also help you to know whether the trim surfaces are correct

## Manufacturing

- In Flatten Tool Surface, the flat pattern with true orientation for curved surfaces is now displayed
- In surface allocation:
  - natural\_path\_method for allocation has been added
  - cutpiece neutral file has been updated
- In Draping, the algorithm has been improved
- In Nesting, the following operations are available:
  - Contour nesting (best match and slide)
  - Sequential positioning and sequential nesting from flat cutpiece data (the cutpieces generated through flat allocation/surface allocation and draping are possible)

- Mixed/fabric/multipart nesting is possible with different tape lengths and different materials
- Generating a nested cutpiece neutral file

## Analysis

- In Define Local Region, select four curves (instead of elements).
- To generate an FE data file, the following options are available:
  - Displacement Boundary Condition: The finite element model of the clipped region uses the displacements along the mid-nodes of the edges as the boundary condition.
  - Forced Boundary Condition: The boundary condition along the edges of the clipped region is the stress state obtained from the global analysis. The clipped region also uses the transverse loads, if any, on top or bottom surface as the boundary condition.

Please note: The old analysis database(\_dif.bin) and \_fem3d.dat are not valid in this version. Reread and recreate the files. Currently, Delamination Analysis is not operational.

# Concurrent Assembly Mock Up (CAMU)

## CHANGE ASSEMBLY

Allows for the revision of the associated assembly to which the selected assembly name is to be associated.

## CONSTRAIN COMPONENT

You can now use the `CONSTRAIN COMPONENT` command to modify the offset value of the selected constrained position.

For constrain types `Mate` and `Align`, the value is displayed in drawing units. For constrain type `Orient`, the angle value is displayed in degrees.

You can now constrain a node of an active reference assembly.

## CV\_ASSEMBLY\_NO\_MODEL

Use this new environment variable to add a component without associating a model with it.

## EXPORT ASSEMBLY

You can now maintain different versions of same assembly.

## MIRROR COMPONENT

Using the `GRAPHIC` option, you can now use the same model graphics for the mirror copied component.

Use the `PREFIX` option while doing the mirror copy or mirroring the component across assemblies to resolve naming conflicts.



# CVACT

## IGETFOCUS

Use this function to get the current focus state of the fillin field.

## UIPARAGRAPHINSERTTEXT

You can insert specified text at any location of the cursor in the paragraph using this function.

## UIPARAGRAPHMOVECURSOR

Use this function to move the cursor in a paragraph in a manner specified by the direction string.

## UIRESETCAB

The `UIRESETCAB` function now resets the entire command box. All the component fields of the command box also reset their value.

## CV Hull

### CVMAC GENERATE STIFFDRAW, PLATEDRAW, and SHIPDRAW

Pitch marking has been enhanced for the `GENERATE STIFFDRAW`, `PLATEDRAW`, and `SHIPDRAW` commands.

The Plate and Stiffner Drawing Generation property sheets have been updated with a new option, Draw Name Suffix. Use this option to specify multiple drawing names.

### PUT NESTFILE

A new modifier, `DATUMNAMES`, has been added to the `PUT NESTFILE` command.

### SELECT SHIPPARAMETER

The `DATUM` modifier and its submodifiers have been added to the `SELECT SHIPPARAMETER` command.

The Plate Nesting File Generation property sheet has a new option, Output File Suffix4. Use this option to specify names for new multiple IGES files.

# CVMAC

## DATE4Y

`DATE4Y` is a new CVMAC command that retrieves the current system date and time. It is similar to the `DATE` command, except that the year value printed is in four digits. The format of the output of the `DATE4Y` command is `YYYY-mm-dd`.

### Syntax

```
<date_stmt> ::= DATE4Y <tvar> {,<tvar2>}
```

### Where

`<tvar>`—Receives the year, month, and day in the ten-character format:

YYYY-MM-DD

`<tvar2>`—Receives the hour, minute, and second in the eight-character format:

HH:MM:SS

# CVNC

## CVNC Legacy Data - Phase I

- Command locking/unlocking
- Toolpath comparison and toolpath recovery

### LOCK/UNLOCK

Use the `LOCK` command to lock JCF's from executing so that the original toolpaths produced on earlier versions of the software are retained.

Use the `UNLOCK` command to unlock JCF's or command lines that have been locked using `LOCK` or if you have entered the JCF with the `LOCK` option.

Unlocked JCF's or command lines can potentially re-execute.

### FC

Use the `FC` (Forward Compare) command to evaluate the new toolpath (if any), with respect to the old toolpath, by displaying both graphically or/and in text form any differences found. You can then decide to keep the new toolpath or old toolpath, based on your evaluation of the differences.

### RECOVER

Use the `RECOVER` command to recover the old toolpath, in the event you decide to retain the older toolpath after an `FC` operation.

### COMPARE

Use the `COMPARE` command to set the color, output options and tolerances for the `FC` command.

## NC Builder

All 3- and 5-axis commands are now available within NC Builder.

The following tasks in the NC Builder Task Selector have been enhanced:

- `STRAIGHT`, `CURVE`, `ZPROFILE`, `PROFILE5`, `INTERSECT5`
- `MPOCKET`

- SURFANGL with two STRAIGHT or CURVE tool paths
- SURFANGL with a STRAIGHT and a ZPROFILE tool path
- SURFANGL with a CURVE and a ZPROFILE tool path
- SURFINT3 AUTO
- SWARFCUT EDGES

## PROFILE

Two new modifiers, CLW and CCLW, have been added to the FACE option. CLW or CCLW with FACE allows you to select the face entities in the clockwise or counter-clockwise directions, respectively.

Two other new modifiers, STARTPT and ENDPT, allows you to trim the tool path at the beginning and/or end, respectively. STARTPT is mutually exclusive with STARTLIN. ENDPT is mutually exclusive with CHECKLIN.

Please note: CLW and CCLW can only be used with the FACE modifier and are mutually exclusive. When you use FACE, CVNC selects the LEFT and RIGHT sides according to your choice of CLW or CCLW.

You can use STARTPT/STARTLIN and ENDPT/CHECKLIN with or without FACE, as explained next.

### With FACE

In this case, STARTPT defines the starting entity and the tool path direction is based on your choice of CLW or CCLW.

### Without FACE

In this case, STARTPT and ENDPT define the tool path direction, if you have specified both. However, if you specify only one of them, the direction is based on the order in which you have selected the entities and starts or ends at the STARTPT or ENDPT, respectively.

Use the REV modifier if you select the STARTPT and ENDPT (and thus the machining direction) in an order opposite to the order in which you have selected the entities.

Please note: If you do not specify STARTLIN, CHECKLIN, STARTPT, or ENDPT, the cutter machines the entire length of the input boundaries and you do not have control over the start and end points (for the FACE option).

## SURFINT3 AUTO

You can now specify an even number of cuts while using the `NUMCUTS` modifier. In this case, the pair of corner cuts are made at half the stepover distance from the corner and the remaining cuts are separated by the stepover distance.

## SWARFCUT and PREPARE CURVSURF

A new modifier, `EDGES`, has been added to the `SWARFCUT` command.

In `EDGES` mode, surfaces (including trimmed surfaces) are machined by aligning the tool axis to the edges and surface-to-surface transitions of a curve surface created by the `PREPARE CURVSURF` command. You can also specify two rails to position the tool axis using `PREPARE CURVSURE`.

# Design and Drafting

Context-sensitive dimensioning is new in this release.

## INSERT DIMENSION

You can use this command to create:

- Linear dimensions—Horizontal, Vertical, Point to Point Parallel Perpendicular
- Radial and Diametrical dimensions
- Angular dimensions

Scrolling, Zooming and Repainting is also supported.

## MODIFY DIMENSIONS

- Modifying properties -
- Modifying a single arrow
- Modify Text Properties
- Aligning Linear and Angular Dimensions

## Associative Centerline/Label/Hatch to Viewed Parts

You can now associate centerline/label/hatch to viewed entities. Viewed entities become selectable for associating centerline/label/hatch with them.

The centerline/label/hatch associated with the viewed entity can be made non-associative by using the `CHANGE DIMENSION NONASSOCIATIVE` command

The following commands have been enhanced to pick viewed entities and associate centerline/label/hatch.

- `INSERT CLINE`
- `INSERT LABEL`
- `INSERT HATCH`

## REGENERATE DIMENSION

You can now pick the centerline/label/hatch associated to viewed entities and processes them.



## EPD Enabled CADD5

This is a new functionality enhancement for CADD5 revision 9.0. on Solaris and IBM operating systems, which provides:

- CADD5 as a Life Cycle Application in EPD.Connect 3.0.
- New look and feel (Menu changes + context sensitive object actions)
- New way of Parts & Assemblies Management
- EPD.Connect InfoBrowser as Data Manager, in place of LDM.
- EPD.Connect Common Open Dialog inside CADD5 desktop (Provides access to organization's data sources and EPD.Connect services inside CADD5 desktop)
- Features/Properties User Interface inside CADD5 desktop
- New intelligent behavior during Part activation
- Creation of new Parts/Assemblies from EPD.Connect

EPD Enabled CADD5 is a separately licensed module

# Explicit

## FIND DISTANCE

Use this command to determine the distances, both minimum and maximum, between pairs of geometric entities or two groups of geometric entities. The geometric entities can be:

- Points
- Curves
- Surfaces
- Faces
- Trimmed surfaces
- Solids
- Strings
- C-points
- G-points
- M-points
- S-points
- N-lines
- C-nodes
- Text-nodes
- Nfigs
- Nodal-Texts

You can also use `FIND DISTANCE` to determine the minimum and maximum distances between two groups of entities when there is a solid or trimmed surface in either of the groups. `FIND DISTANCE` also determines the minimum distance between a solid and a group of non-solid entities.

`FIND DISTANCE` calculates the maximum and minimum distances when:

- Intersecting entities from different groups are not a closed solid.
- Intersecting entities from different groups are a closed solid.

In this case, the minimum or maximum distance determined is the distance from points of entities of one group to the nearest shell of that solid, and not to the nearest point of the solid.

`FIND DISTANCE` determines whether or a given point of an entity is within a closed solid, though there is a degradation in the performance of the command.

The default is to find the minimum distance between pairs of entities or groups of entities.

## INSERT PART

This command has been modified so that you can specify the list of layers from the original part containing entities you want to insert into the currently active part.

To do this, use the `SRCLAYER` modifier and its sub-modifier `DISSOCIATE`. The `DISSOCIATE` modifier is available only if the `SRCLAYER` is selected.

## DRAGGING PROJECT

The improved mechanism of dragging geometry dynamically solves constraints. The mechanism is basically drag and drop using the mouse.

## PARAMETRIC SKETCH/LOFT (a prototype)

The Parametric loft option, `INSERT SKLOFT`, automatically invokes Sketcher. In this mode, you can use construction entities in Sketcher for geometry that should not be applied to the lofting operation in the Parametric environment.

## UNIFIED DIMENSIONING

A single option called the Unified Dimension Tool (UDT) facilitates dimensioning. The cursor exhibits context sensitivity and rubberbanding behavior. You can select objects dynamically. The type of object you select prior to dimensioning and the location of the cursor relative to the selected object decides the type of dimension.

## OBJECT SEWING

You can sew together a series of open objects which form a closed loop into a single closed object. The closed object can be assigned a single object sense and can participate in autotrimming.

# Explicit NURBS

## APPROXIMATE NSPLINE

You can use the new `APPROXIMATE NSPLINE SAMEKNOT` option when you want the approximated curve to have the same number of knots as the reference curve.

Please note: Use `SAMEKNOT` only when reparameterizing a curve using the knot vector of another curve or surface.

## EXTEND NSURFACE COMPLEX

This command now includes two new options:

`ACCURATE SURFACE` specifies that the extended surface, by default, is accurate for normal, as well as complex, triangular surfaces.

`ACCURATE EXTENSION` specifies that the extension length is accurate, especially for complex triangular surfaces.

By default, the extension you create does not have the exact length specified.

## FILLET NSURFACE

You can use a new `FILLET NSURFACE` modifier, `WOPTIMIZE`, to create constant and variable radius fillets. You can also specify that you do not want data reduction of the filleted surface. This speeds command execution, especially for complex surfaces.

The icons for the options "Variable radius fillet using a guide curve" and "Variable radius fillet using the law of variation" on the Fillet Nsurface property sheet were similar. The icon for the option "Variable radius fillet using a guide curve" on the Fillet Nsurface property sheets has changed. This affects seven menus of `FILLET NSURFACE`. This change is only a menu change and does not affect the functionality.

## INSERT FLANGE

`INSERT FLANGE` now works on a series of curves or a set of surfaces.

# Graphics

## Customizer Icons

You can now customize existing CADD5 items or icons using the Customizer menu icons.

## Edit Icon

You can now customize any CADD5 blank item using the Edit icon within the current CADD5 session. To display the customized blank item each time CADD5 is started or when you are moving to a higher CADD5 version, save the customized blank item with the Save icon.

# Harness Layout—Parametric

## GENERATE SHAPE

A new command, `GENERATE SHAPE`, allows you to create a surface or solid display representation of selected channels in the model using the cross-section related attributes defined on the channel.

## INSERT CONNECTPOINT

A new modifier, `CENTER`, has been added to the `INSERT CONNECTPOINT` command. Use this modifier to specify the location of the center connection point.

## ROUTE CHANNEL

The following enhancements have been made to `ROUTE CHANNEL`:

- Provides the capability for a new channel to follow the geometry of an existing channel, maintaining associativity with the existing channel
- Supports round, rectangular, elliptical, round rectangular, and filleted rectangular cross-sections during channel routing
- Provides the capability to route a channel along a curve or surface

The following new modifiers have been added to the `ROUTE CHANNEL` command:

- `SHORTESTPATH`—Specifies that the channel is to be routed along the shortest path between two components, satisfying minimum bend radius, tangency, and direction conditions defined on the components.
- `DISTANCE`—Allows you to create a channel branch at a specified distance from the channel end or harness component.
- `COMPONENT`—Allows you to specify the branch position at a fixed distance from any harness component, such as a clip, connector, or junction component.

## Other Enhancements

- `EDIT CHANNEL` and `SELECT HDATTRIBUTES` commands have been enhanced to allow you to change cross-section profile attached to the channel.
- `INSERT HDCOMPONENT` now allows you to insert break connectors in the Parametric environment.
- `EDIT HDCOMPONENT` allows you to edit break connector components.

- **MODIFY CHANNEL** has been enhanced to allow you to insert a sag between supported sections of a harness segment, considering the effect of gravity.
- You can now associate a text marker with a channel.

## Harness Layout—Explicit

A new option, **CHAIN**, has been added to the Create Channel property sheet. This option prompts you to select a line/curve for channel insertion.

All the connecting lines/curves between the two Cnodes are selected automatically. The selected line/curve must lie between the two Cnodes.

# ISD

There are two new ways of creating ptons on a curve:

- Create a pton on a curve at the end point which is closest to the point of selection on the curve
- Create a pton on the curve at a point equidistant from both the ends of the curve

There are two new ways of creating ptons on a surface:

- Create a pton at a specified U,V parameter value on the surface.
- Create M number of ptons on the surface along the U direction of the surface and N number of ptons appear along the V direction of the surface.

There are five new ways of creating a data set-

- Create a data set from a curve and include the tangency and curvature constraints.
- Create data sets from a surface either along U direction or along V direction and include the tangency and curvature constraints.
- Project a data set on a surface or a shell and the result is a 3D data set.
- Create a data set from a curve on surface or a polycurve and include the tangency and curvature constraints.
- Specify locations on the surface or the shell, the created data set lies in the U,V parameter space of the surface or the shell.

Now you can create polycurves in ISD as follows:

- Project a curve or polycurve on a surface or a shell and the result is a polycurve.
- Create a polycurve on a surface or a shell by using:
  - Projected curve
  - Iso curve
  - Section curve,
  - Smoothing through a data set on a surface or a shell
  - Interpolating through a data set on a surface or a shell

Creation of on-shell entities is new in ISD for this revision. You can create polysurfaces (shells) in ISD with polycurves as input.

You can perform a following operations on polysurfaces:



- Refine the mesh
- Translate the polysurface
- Project a curve, a data set or a polycurve onto a polysurface

Some new surface analysis tools are now available:

- You can create a plane set and use it for sectioning surfaces or polysurfaces.
- You can create section curves by intersection of plane sets and surface or shell.

Use these tools to analyze the quality of surfaces and shells.

You can compare surfaces along the common edge by reporting the distance and angle of deviation at the specified locations. You can also measure distances between two entities (curve or surface).

# Parametrics

## CHECK DBASE

Note the following new recommendations for using the `CHECK DBASE` command:

- If you are unsure of the validity of a command result, use the `Valid` option. (Use this option only for diagnostic purposes, as it does not modify the database.)
- To remove orphaned entities from the parametric database (`_fd`), use the `Clean` option.

## EDIT PPART

The `EDIT PPART` command allows you to edit the existing `INSERT PPART` command. You can add, remove or replace the entities from the selection as specified previously by using the `INSERT PPART` command. In addition, you can propagate changes done in the source part to the target part.

## INSERT BEND

You can now insert bends on all the vertices of the selected pipeline

## INSERT FILLET

The `INSERT FILLET` command has been enhanced to create a fillet between wire-frame entities or curves, using combination of:

- Two curves and radius of fillet
- Three co-planar wire-frame entities
- A curve, a location, and radius of fillet

## PLOT DRAWING/PUT CGM

The `PLOT DRAWING` and `PUT CGM` commands have been enhanced to plot the zoomed portion of the drawing magnified (as it appears on screen), using the `Aszoomed` option.

# Parametric Piping

Parametric Piping includes the following new commands.

## EDIT PIPE

Use this command to edit the attributes of an existing pipeline. You can edit pipeline attributes, such as size, specification, mnemonic, line number, and change the layer of an existing pipeline.

You can

- Show the flow direction of a pipe series.
- Reverse the flow direction of a pipe series.
- Change the pipeline layer by specifying the layer name or number.
- Terminate a pipe series by selecting a location for termination.

## GENERATE EXPLICIT PIPELINE

You can generate an equivalent explicit pipeline for a parametric pipeline that is routed using the pipe routing functionality. This functionality is available to you from the explicit environment.

## GENERATE PIPELINE DATABASE

This command allows you to generate and store the pipeline data in an external relational database for specified pipelines. You can append, overwrite, or delete the existing information. In addition, you can execute external script files for generating reports. You can specify all the pipelines in the active part, or specify them by the mnemonic and number.

## GENERATE SHAPE

You can create surface or solid display entities for pipelines with the specified mnemonic and specified numbers. You can select a pipe series or all pipelines for surface or solid display representation.

## INSERT BEND

Use this command to insert bends at specified vertices of a selected pipeline. You can insert bends by specifying the bend radius, or bend ratio, or by automatically selecting the bend radius from the external RDBMS tables based on the pipeline size and specification.

## INSERT FITTING

Use this command to insert fittings on a selected pipeline. You can insert general fittings, branch fittings, elbow fittings, or reducer fittings. You can also insert fittings at a specified location in free space.

In addition, you can

- Insert an inline fitting and orient its local y-axis along the global co-ordinate axes using the UpipeCpl option.
- Rotate the fitting clockwise around its local x-axis.
- Flip the local x-axis of a reducer fitting.
- Insert a fitting face-to-face at the end of an existing fitting.
- Offset the fitting by specified distance from the selected location.

In addition, you can

- Choose a fitting selection name from a list. A shaded picture of the corresponding library part appears on the right side of the list. You can scroll through the list, view the corresponding icon, and select the required fitting from this list.

## INSERT PIPELABEL

Use this command to:

- Display the label of one or more pipelines at any given location.
- Display the labels of all pipelines in the active CADD5 part.
- Suppress the label of one or more pipelines at any given location.
- Suppress the labels of all pipelines in the active CADD5 part.

## MAKE PIPE

Use this command to display a pipeline using surface or solid representation. The surface or solid entities created by this command are placed on the currently active layer. If the shading option of core Parametric CADD5 is enabled, the display entities are automatically shaded. The display entities are associated with the corresponding pipeline.

## REPORT PIPE

Use this command to report pipeline objects. You can report pipeline segments, pipeline bends, and pipeline fittings.

You can also report all the pipeline networks in the active part and report pipelines by specifying the mnemonic and number.

## ROUTE PIPE

Use this command to route pipelines:

- Based on the location, entity, as well as free space routing.
- From equipment nozzles and components.
- Between two selected lines at an equal distance or along two selected curves.
- To or from the start or end of an existing pipeline.
- Along the specified axes of the active Cplane between two selected locations with locking of the axis.

In addition, you can

- Add a vertex to a selected pipeline.
- Delete a selected vertex from a pipeline and route that pipe segment without the vertex.
- Route pipe segments that follow a selected pipe segment, pipe series, or a part of the pipe series specified by the start and end segments.

In addition, you can route a pipe segment:

- Of the shortest length between two selected entities.
- Tangent to selected curves.

## SELECT PPARAMETERS

Use this command to:

- Specify default modeling parameters in the active CADD5 part. This includes specifying the default fitting library and pipeline label format.
- Connect to and disconnect from the external relational database.
- Specify a project database for access by the RDBMS-related Parametric Piping commands.
- Display certain modeling parameters in a read-only window.
- Report all the selected default parameters in the report window.

## Piping Enhancements (Pipe Supports)

### PS\_FAB\_STAGE\_<project\_name> Table

The <project\_name> varies for different projects, where each project will have its own set of tables, except for MDF table, which is used for all projects.

### PS\_UNIT\_<project\_name>Table

The values of the `SUPPORT_FRAME`, `SUPPORT_LONGITUDE`, and `SUPPORT_DECK` fields in the unit table are automatically obtained during the execution of the `GENERATE SUPPDBA` command by checking the location of support against the following file, if it exists.

```
/cadds/src/data/cvaec/pipesuppor/hcf
```

If the file does not exist, a null value is assigned to the above fields in the unit table for the support unit. `SUPPORT_ZONE` and `SUPPORT_BLOCK` are retrieved from the `ZONE` and `BLOCK` property in the support parent `nfig`. Use the `INSERT PROPERTY` command to insert `ZONE` and `BLOCK` properties.

# Sectioning

## DEFINE SECTION

Some of the HLR options are now available when you select the CUTAWAY option.

## REGENERATE SECTION

The ALL OBJECTS option has been added to the REGENERATE SECTION command. You can regenerate the sections for all the solids that are currently viewed, using the All Objects option from Regenerate Section menu. This results in all the viewed solids to be considered as sectionees during regeneration, irrespective of whether they were sectionees during the Define Section operation.

## UNDEFINE SECTION

Use the new ALL option for this command to undefine all the options in the current drawing.



# Sheet Metal Design

## BENDALLOW and BF

Two new options, `AFON` and `AFOFF`, have been added to these commands.

- `AFON`—Allows you to select individual vertices to turn Auto Fillet on, overriding the global variable setting.
- `AFOFF`—Allows you to select individual vertices to turn Auto Fillet off, overriding the global variable setting.

## Bend Allowance Global Variables

When setting global variables for Bend Allowance on the Bend Allowance Global Variables property sheet, you can now define a User-Defined Constraints equation on the property sheet.

Please note: You can also use the `ADD EQUATION` option on the Constraints task set to add an equation.

## Translators

The command name for activating the Datashop UI has changed from `cvuts_gui` to `datashop_gui`.

The Datashop User Interface has been generalized to support all the translators: AP203, AP214, IGES, VDA and CAMU IGES.

The "IGES GUI" and the "Translator GUI" has been replaced by "DataShop".

You can use the Datashop user interface for application protocols such as AP203, AP214, IGES, CAMU IGES, and VDA-FS.

Using the Datashop, you can easily change the translator options and also translate multiple files, with the same modifier configuration. You can still use the translator commands directly from the system prompt, without accessing the user interface. Use the `datashop_gui` command to access the datashop.