

What's New in CADD5[®] 5 Revision 8.0

CADD5 5 Revision 8.0

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Preface

What's New in CADD5[®] 5 Revision 8.0 is an introduction to what has been added to CADD5 5 since CADD5 5 Revision 7.1.

Your Comments

We welcome your suggestions and comments about this or any other Computervision documentation. You can send electronic mail to the following address: `doc_input@cvbnet.cv.com`.

Online User Documentation

Computervision supplies online documentation in HTML and provides PostScript files of each book in the online collection. You can print the PostScript files for hard copy 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 8.0*:

- *CADD5 5 Revision 8.0 Tips and Considerations*
- *CADD5 5 Revision 8.0 Release Notes*

Book Conventions

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

Use of the Convention	Examples	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 ₁ 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 8.0

This document provides an overview of what is new in CADD5 5 Revision 8.0. For detailed information, please refer to the product-specific online documentation.

New Products

Interactive Surface Design (ISD) Surface Functionality (Model Number CV3079)

The following functionality is available in CADD5 5 Revision 8.0:

- Surface creation – insert, net, shape, tangent field construction, with history, uv constraints, constant, file input, cycle selection
- Surface modify – drag onsurf, drag normal, tangent field construction, knot handles, knot point string handle, with history, uv constraints, constant, file input, cycle selection
- Surface analysis – planes, normal field, refine, curvature in iso curves, reports on properties
- Curve smooth enhancement
- Reports – curve enhancement
- Curves project – modify display on cpl interactive
- Curve analysis – multi-curve

A new command, EDIT ISD SESSION, allows you to change the range of ISD commands in the part history.

5 Axis Riveting (Model Number CV3557)

This includes the following enhancements:

- NC Builder interface with all appropriate set-up information concerning cpls, single definitions of tool definition to be used for display and visual collision detection
- Point selection through attribute filtering, priority being diameter and type
- Point selection with ordering left/right path generation
- Point “snapping” to the nearest surface or face of a solid
- Degouged route between neighboring points
- All necessary output for specific riveting machine at British Aerospace

Major Enhancements in CADD5 5 Revision 8.0

Several aspects of CADD5 have been enhanced in Revision 8.0. These enhancements and considerations are grouped and described by product.

EPD.Connect

CADD5 5 Revision 8.0 is compatible with Optegra 2.0.2.

INSERT/CONSTRUCT PART

The following features are included:

INSERT PPART

This new command allows you to import all or selected geometry from an existing (source) part to an active (target) part. It imposes a uni-directional relationship between the source and the target parts.

UPDATE PPART

This new command allows you to propagate the changes of a source part to the target part.

DISASSOCIATE PPART

This new command allows you to remove the link between the source and the target part.

Tagging

The Synonymous tags project addresses parametric selection issues, resulting from tag changes due to topological changes and from addition/removal or redefinition of construction steps. Commands that are supported by this project on Revision 8.0 are:

- APPLY DRAFTANGLE
- MERGE FACE
- MOVE FACE
- SPLIT FACE

- INSERT HOLE
- INSERT BOSS
- INSERT POCKET
- INSERT SUBPOCKET
- INSERT SLOT
- INSERT NOTCH
- INTERSECT PROFILE
- SUBTRACT PROFILE
- UNION PROFILE

The Construction Graph project addresses issues related to the selection of top-level results of commands that produce more than one result. This project does not address sub-entity selection issues related to some of the affected commands. Commands that are supported by this project on Revision 8.0 are:

- BREAK PCURVE
- EXTEND FACE
- DROP ENTITY
- INTERSECT ENTITY
- OFFSET CURVE

Integrated 3-D Sketcher

Enhanced Sketch Deletion

You can now delete multiple unapplied sketches or legacy profiles using the Delete Sketch property sheet. Previously, you could delete only one sketch or profile at a time.

There is now a Delete option panel on the Delete Sketch property sheet. Previously, the selected sketch or profile was immediately deleted when you selected its name from the runtime list.

As in earlier revisions, selected sketches and profiles are not actually deleted until you file the active part.

Additional Object Creation Methods

You can now create Nsplines and ellipses. You can also create fillets.

Nsplines: You can dynamically create and modify third degree Nsplines through interpolation points in Sketcher. You can constrain objects to be tangent to Nspline end points.

Ellipses: You can dynamically create and modify ellipses in Sketcher.

Fillets: You can create a fillet at any visible or implicit vertex. A resultant dimension enables you to change the fillet radius at any time. The vertex hotspot remains available for modification.

Diametrical Dimensioning

When you dimension to an arc, a radial dimension is created. When you dimension to a circle, a diameter dimension is created. When you dimension to a circle that has been autotrimmed to an arc, a radial dimension is created.

Construction Entities

You can create construction entities by applying a construction attribute to selected sketch geometry. Construction objects reside in the sketch but are not visible in the model. These enable you to create additional constraint relationships in the sketch.

Copy and Paste

You can select one or more sketch objects to copy and paste in the active sketch. This provides a convenient way of grouping and moving objects. Constraints and dimensions that exist in the object selection group are copied.

CAMU

Parametric Positioning

Parametric Positioning allows you to position components by defining positioning constraints (such as mate, align, orient, insert) so that the positions and orientations of components are inferred. Therefore, when a component moves in an assembly, the new positions of affected components can be recalculated so that the positioning constraints are satisfied.

Three new commands facilitate this enhancement:

- SELECT ASSEMBLE
- CONSTRAIN COMPONENT
- ASSEMBLE COMPONENT

The new functionality is intended to augment the inter-part constraints capability provided in CAMU and PMD. It can be used only from the Parametric Assembly environment.

Parametric and Explicit Positioning: Hybrid positioning is supported for existing and new assemblies. An assembly can have a combination of explicitly positioned components and parametrically positioned components. All existing assemblies can be used without any conversion.

Concurrent Visibility

CAMU Status Window: When a user selects a component in the CAMU tree structure window, the information about the structure displays in a separate window, the CAMU Status window, which appears just above the CAMU tree structure window. The CAMU Status window always displays and cannot be closed by the user. It contains the following information:

- User who has locked the structure
- Model attached to it
- Status of the model (activated, suspended, or does not exist)

LIST ADRAWING: This command has been enhanced to display the name of the user who has write permission to a particular Adrawing.

VERIFY OBJECT: This command generates an extra field of model status in the listing.

Referenced Assemblies

You can now activate and edit a reference assembly from within the active main assembly.

You determine the maximum number of reference assemblies that can be open in an active main assembly by using the new environment variable CV_MAX_ACTIVE_REF. By default, at most three reference assemblies can be active at the same time in an assembly session.

When the limit of active reference assemblies is reached, you can use the new command `QUIT REFERENCE` to deactivate an active reference assembly.

This enhancement enables you to:

- Write to a reference assembly database (`_db`) using commands such as `ADD COMPONENT`, `REMOVE COMPONENT`, `CHANGE COMPONENT`, and `REORDER COMPONENT` from within the active main assembly.
- Activate, edit, and file a model that is associated with a component in an active reference assembly using `ACTIVATE MODEL`.
- Suspend a previously activated reference assembly, enabling you to iterate modifications on several referenced trees before making validation decisions (filing).
- Lock components in a reference assembly to perform commands such as `ACTIVATE MODEL`, `ADD COMPONENT`, and `ASSOCIATE AGROUP`.

Please note: Locking any component in a reference assembly activates that reference assembly in the active main assembly.

Please note: `REFRESH ASSEMBLY` can detect reference assemblies that have changed in the current session but have been deleted by another user working concurrently in the same main assembly. This opens an alert window with a prompt for filing or quitting the main assembly.

A new command, `EXPORT ASSEMBLY`, is also available.

General Assembly Database Enhancements

General assembly database functionality has been significantly enhanced. You can now:

- Create a new assembly database by exporting a subassembly.
- Optionally convert a subassembly to an instance of a reference assembly.
- File the active assembly as a new name.
- File an active or suspended model as a new name.

Implicit Auto Lock

This function automatically locks all components that must be locked to perform a particular command. You no longer need to use the `LOCK COMPONENT` command before modifying an assembly.

The locked component still displays an asterisk (*). The key difference between an implicit lock and a regular lock is that the implicit lock is released and the regular lock remains when you file the assembly. When the implicit lock is released, the asterisk symbol on the implicitly locked component disappears.

Graphics & Detailing

Assembly Drawing Associativity

Associative Dimensioning to Viewed Parts: Dimensions can be associated with viewed parts, either at the part level or assembly level. Also, when a viewed part is changed or relocated, the attached dimension is automatically updated.

Associative Dimensioning to HLR Entity: Dimensions that are associated with HLR limbs/surrogates are preserved when the model or assembly component is modified. UPDATE HLRIMAGE preserves the previously created dimension wherever possible. Similarly, HIDE OBJECT, if applied to previously HLRed entities, also preserves dimensions of limbs/surrogates.

Associative Dimensioning to Sections: Dimensions that are applied to a section view (DEFINE SECTION) maintain associativity when changes are made to the sectionee and sectioner. After changes are made, regeneration of sections preserves the previously created dimensions wherever possible.

Associative Section Crosshatching: DEFINE SECTION automatically creates associative crosshatching. Sections through multiple solids (assemblies) have opposite crosshatch angles for adjacent solids. User-made changes to crosshatch parameters are maintained during regeneration of sections.

Context Sensitive Dimensioning (CSD)

CSD Change Dimension allows you to identify and change individual components of a dimension. The appropriate options for changes display, depending on the dimension type. Current dimensional values display in a tabulated dialog box for dynamic changes. In addition, CSD Change Dimension allows you to relocate dimensions via dynamic mouse dragging. When CSD Change Dimension finishes, CADD5 returns to its normal state with all its functions available.

The following dimensional components are available for change to LDIM, DDIM, RDIM, and ADIM:

- dimension text

- dimension lines
- dimension extensions
- dimension arrowheads

PUT VRML

CADD5 surface and solid geometry can be output to Virtual Reality Modeling Language (VRML) for distributing 3-D images across the Internet. VRML is a modeling language for specifying interactive animation.

VRML is a platform-independent format that allows the display of 3-D model assemblies through the Web in full color with special texture, animation, and lighting effects.

Auto Update of CADD5Shade

CADD5Shade files are automatically updated when you file them if the CV_FILE_SHADE variable is set. Shade files will be created if necessary.

CV-DORS

The following new functions are available:

- New method for the t_surface class, which allows you to project a point on a surface along a given direction
 - t_surface::project_point_dir is the C++ method name
 - t_surf_Project_point_dir is the C callable interface
- New method for the t_face class, which allows you to create a trimmed surface out of a face
 - t_face::get_tsurf is the C++ method name
 - t_face_Get_tsurf is the C callable interface
- Enhancement to the t_brep sewing constructor, which allows you to control the tolerance
 - t_brep::t_brep is the C++ method name
 - t_brep_sew_var is the C callable interface
- Addition of a new flag to the existing t_brep trim functionality, which allows you to suppress the joining and subdivision of an input trimming contour
 - t_brep::trim is the C++ method name
 - t_brep_trim_without_join is the C callable interface

- New method for `t_entity` class which allows you to get an entity color
 - `t_entity::get_color.3` is the C++ method name
 - `t_entity_get_color_index.3` is the C callable interface to get color index information for layers
 - `t_entity_get_color_rgb.3` is the C callable interface to get the color rgb value information for the specified entity
- New method for `t_db` class which allows you to return a color for the specified layer
 - `t_db::get_layer_color.3` is the C++ method name
 - `t_db_get_layer_color_indx.3` is the C callable interface to get color index information for layers
 - `t_db_get_layer_color_rgb.3` is the C callable interface to get color rgb value information for layers

Manufacturing & Analysis

NCBuilder Turning

New functionality is available for profiling and discrete tool motion, such as absolute motion, incremental motion, location motion, entity motion, and XYDir entity motion.

New functionality is available for turning, facing, and four-axis synchronous machining operations. Drilling, tapping, and threading can also be performed using the NCBuilder Turning GUI.

Please note: This functionality requires a license for NCBuilder.

Toolpath Generator for Turning and Facing

This includes new tool path generators and adds the ability to perform four-axis (2x2 axis) turning, including the following:

- Control of machine spindle direction
- ANSI standard tooling definition support
- Control over extension of start/end check curves
- Automatic 2D material recognition and updating
- Clearance angle on back angle support

- Cut direction control with default from current tool drive vector
- Step in/out vector control
- Start point control, either explicit or derived from material boundary
- Optional return to start point between passes
- Cut depth control by maximum depth or number of cuts
- Control of material left for optional final profile pass incremental end offsets to prevent tool tip from being buried in material from previous cut due to tool insert shape
- Global and differential stock support (per entity)
- TO/ON support of check curves
- Start and end tool path extensions
- Optional final profile pass
- Maximum undercut depth control
- Clearance angle on back angle support

3 Axis Enhancements – Rougher, Phase II (High Speed Machining, Phase II)

This includes the following:

- Steppoff distance option
- Loop machining option
- Multiple boundary pass option
- Wireframe part option
- Material offset option
- Leadin/leadout with ramp

5 Axis Enhancements – Intersector

This project stabilizes existing corner machining and Surfint5 functionality, fixes quality issues reported against Surfint5, and implements optimal cut ordering in corner machining.

NCBuilder 5 Axis

This project provides NCBuilder style tasks for each of the following CVNC M5 commands:

- SURCUT5 STRAIGHT
- SURFCUT5 CURV
- PROFILE5
- SURFINT5

These new NCBuilder tasks have been added to the NCBuilder task selector and are available only in CVNC M5.

Enhancements have also been made to the NCBuilder Setup block to support machine tool configuration and multi-axis output commands. These are also available in CVNC M3, since they are the same in that product. The support for machine tool configuration is limited to the support of the current CVNC command CONFIG. No enhancements to the core CVNC machine tool definition have been made.

NCBuilder milling (3 and 5 axis) have been enhanced to support a text tool library in addition to the existing CVNC TLIB. This will be done in a similar way to the insert/holder/box tooling file implemented for NCBuilder Turning.

SMD 3-D Features

By turning a toggle switch (in folder global options) you can see a 3-D actual representation of the feature "booleaned" with the folded solid for the same feature. Therefore, you have a handle to toggle between the 2D and 3-D representation in the folded model.

Ship/Plant/Electrical

Routed Geometry, Phase II

The Routed Systems project, through a new command, **MODIFY CHANNEL**, allows you to insert drip loops and service loops on channels and to check the minimum bend radius during channel routing. It provides pre-placed splice functionality and percentage fill constraint capability for wire routing. Segregation constraints are also enhanced. The branch point of a channel is parameterized. The wire path editing capability is refined. The reports and charts generated include the tarelen or additional length specified on the channels.

Service/Drip Loop (Parametric Channel Routing): Drip loops are used to trap moisture. Service loops provide extra wire length for tolerance and servicing. The service loop and drip loop are provided at the user-specified location of the selected channel. An additional modifier is provided and parameters can be set

through the various property sheets to facilitate creation of the service/drip loop geometry.

Check Minimum Bend Radius: The minimum bend radius check ensures that the channel is routed to satisfy the minimum bend radius specified by the user for that channel.

Change Parameter of Branch Point: The branch point is parameterized to allow you to move this branch point using the `CHANGE PARAMETER` command. A new modifier, `TVALUE`, for the `ROUTE CHANNEL` command, allows you to enter a value between 0 and 1, which corresponds to the parameter value of the point on the curve at which the automatic junction is created. This allows you to control the position of the junction after it is created.

Harness Layout: Two properties, `Chanshape` and `Pfill`, have been added to the Parametric and Explicit harness channel.

Percentage Fill Constraints for Rectangular and Circular Cross Sections in the Wire Router: The percentage fill constraint checks and disallows wire routing through channels that are already filled to or beyond the specified percentage.

Rsys Toolbox: A new toolbox has been added to Parametric Harness Layout. The toolbox allows you to insert connection points in a harness part.

Tare Length Implementation in Explicit: You can now add more length to a channel. The additional length is added while reporting and annotating.

Unit Conversion for Menu Fill-In Field Data: New functionality has been added to use units other than the part/drawing units for data input through the menu fill-in fields and command line.

Update Topology (Parametric): `UPDATE TOPOLOGY` is implemented in Parametric as it is in Explicit.

Segregation Constraint Enhancement (Explicit Wire Router): Segregation constraints are enhanced to handle the following:

- Wire must not go through a channel: functionality enhanced to work in all cases
- Wire must go through a channel: new functionality

Improvement in Explicit Wire Path Editor:

- Provides list of nets in all cases

- Provides highlighting improvements, such as highlighting in different colors, only for bend radius check, not for highlighting nets and wires in the router

Pre-defined Splices (Explicit Wire Router): Allows you to place predefined splices in the model that comes from the netlist. Previously, only the generic splices were placed in the model. These splices are considered by the wire router during wire routing.

INSERT SPLICE: This new command has been added, which allows you to insert a predefined splice (one that is defined in the netlist file) in the active harness.

Electrical Cable Modeling Phase II

The following functionality is new:

- GUI for Run Time Lists (RTLs)
- Cable Data Management Module
 - Basic cable reporting
 - Basic cabletray reporting
 - Basic cableway reporting
- Cableway Routing and Cabletray Insertion Module
 - SELECT NEARNODE (new command)

The following commands have been enhanced:

Cable Block Diagram Module

- SELECT CABLEPARAMETERS
- MODIFY CABLE
- DELETE CABLE
- GENERATE EDEVICEDBASE
- INSERT EDEVICE

Cableway Routing and Cabletray Insertion Module

- INSERT EDEVICE
- GENERATE EDEVICEDBASE
- INSERT CABLETRAY
- ROUTE CABLEWAY

- GENERATE CABLEWAYSECTION
- INSERT CBEND
- HIGHLIGHT CABLE

Cable Routing Module

- ROUTE CABLE

CVHull

The following features are included:

- Creation of a plate from multiple surfaces and a defined boundary
- Preservation of local edits to endcuts. Avoid delete and recreation of cutout, collar, endcut during update
- Enhancement of RDBMS Interface
- Enhancement of plate drawing generation command
- Creation of an Stelement on multiple segments of the selected axis entity
- Creation of multiple Stelements on a single twist surface
- Advanced distortion mechanism for clearance cutouts and collars
- Ship classification drawing generation – Phase I
- Reporting of individual weld lengths
- Autogeneration of weld notches
- Panel drawing generation
- Stiffener manufacturing drawing
- Select Shipparameter menu

Advanced Structural Modeling (ASM)

The following have been added/enhanced:

- Edit STobject ALL submodifier added
- Insert STendcut menu
- Insert STElement menu
- Manipulate STobject menu

MCAE/StressLab/PlasticsLab

Parabolic Beams Elements

In addition to the linear element subclass for beams, StressLab now includes the parabolic subclass as part of the element beam table.

Region-Based MESH Paving

The CONTOUR Mesh functions have moved from the Prototype menu to the Region Add panel as a new 2D meshing option. This allows you to create parametric regions that are paved meshed with LINEAR planestrain, thinshell, axisymmetric, or gate elements. Both triangular and quadrilateral element geometries are supported.

Nastran Element Properties

The ATTACH NASTRAN PROPERTIES command allows you to add some optional properties that aren't required for StressLab, but may be applied to elements when creating an FEA model for use in NASTRAN. Some of these properties may be for other types of analytical solutions, such as large deflection or other nonlinear solutions, that aren't common for linear static analysis.

SET DEFAULT ELEM1D

A new function to override the session variable set in the .caddsrc has been added to the SET DEFAULT command. The ELEM1D modifier changes how beam-like elements display in the MCAEDISPLAY window.

The SOLID option means that the beam appears as a cross in StressLab with the Z axis being longer. In PlasticsLab, the solid option makes the element appear as an equilateral cross. Other options are ROD and BOX. For PlasticsLab, Runner and Cooling Channel elements are ROD by default; Connector elements have a triangular crosssection.

PlasticsLab Third-Party Flow Solvers

Moldflow's Flowcheck and CMOLD's Quickfill mold filling solvers are integrated into PlasticsLab. Product activation is made by obtaining a license from the third party.

In addition, you can read and write the FEA models in the native third-party formats with the other plastics analytical solutions from these vendors. This includes cooling, warping, gas-assist filling and packing analysis. Support for reading results for those other solutions is also provided.

Vector Plots

MCAELabs support element vector plots via the MCAEDISPLAY tool. In PlasticsLab, this is used to display melt flow vectors and velocities, or fiber orientations. In StressLab, this is used to visualize element force vectors and stress tensors.

DIF Enhancements

MCAELabs allow you to output only a subset of the FEA model by invoking the selection tool. Any nodes and elements selected also include any attached boundary conditions and properties in the resulting DIF file.

Also, you can renumber any or all of the model using an offset starting number.

ANSYS Rev 5.3 Support

You can format out the FEA model to ANSYS version 5.3.

Lower Priority Nastran Cards

StressLab requires that you assign an element mnemonic table to each element type, assigning it the appropriate Nastran "C card" name. Element properties can now be added to tables to fully define the Nastran data. Nastran physical property tables are now defined as explicit PID cards for the various elements and use Nastran terminology for each field. With these two additions to StressLab, most Nastran elements are now supported. There are a few special elements for which there is no direct StressLab element to relate to, and a few special load entities that need to be supported.

Read/Write Subsets of Data

StressLab can write the Finite Element model information into a neutral ASCII format called a "DIF" file. StressLab writes the entire database to this file when the command is issued. This file is good for archiving an FE model that can be read into another part, or be used as input to other programs.

This new functionality allows a DIF file to be created from a subset of nodes and/or elements. Options include the ability to write out associated properties, loads, and groups.

Vector Plotting

StressLab has several visual displays to help understand the results of a Finite Element analysis. These displays include deformation, contour, quilt, and animation. This enhancement is used to interpret results that have a vector direction associated with the results.

This functionality provides another method for reviewing results generated by a Finite Element analysis. This function works on any vector type data stored in StressLab's post processing database.

Other Applications

CADD5 Composites

The following changes are included:

- Modifications in dialog boxes and user interfaces
- Algorithmic changes for the following commands:
 - CREATE SUBLAMINATE (stagger)
 - CREATE SECTION
 - OFFSET SURFACE commands
- Quadratic failure criteria for GENLAM
- New commands for manufacturing ply editing
- Modifications to ply database

Standalone IGES, Revision 3.1

CAMU IGES: You can output CAMU data in the IGES data format. The user can select a top-level CAMU assembly for processing via the GUI. A graphical tree representation of the CAMU assembly part displays. You can select nodes for processing individual components, sub-assemblies, or the whole assembly. Each node component selected is mapped to the IGES file as an IGES sub-figure (408/308 entity type). Reference assemblies are handled based on any limitations of the CAMU CVDORS access routines.

EPD licensing is incorporated.

Standalone VDA-FS, Revision 3.1

EPD licensing is incorporated.

Standalone STEP, Revision 3.1

Fixes and enhancements have been made to the current functionality for STEP 203. EPD licensing is incorporated.

CADD5 5 Revision 8.0 License Management

This version of CADD5 5 Revision 8.0 requires an updated license file, which you should request from your Computervision Services representative.