

Release Status

Supported CAD Systems, Kernels

Kernels: ACIS R21, CATIA V5 R20.2, Unigraphics NX7.5, Pro Pro/E Wildfire 5, SolidWorks 2010, Inventor 2011, DXF/DWG up to AutoCAD 2010, CADD5i R15 (R16 import on Windows)

Supported Platforms and Operating Systems

Windows 2000, XP SP3, Vista & 7 (32 and 64 bit)
Sun Solaris 8
Linux RHEL 4 & 5 (32 and 64 bit)

Unsupported Platforms

CADfix no longer supports HP HP-UX

Import/Export

Automatic set creation on import based on defined CAD label

Change the import filename by selecting the file then under the Options dialog edit the name. Changing the filename using this option also updates the export filename.

CATIA V4

New import option to preserve SOLIDM as CADfix faceted solids
New export capability to write SOLIDM entities
New export option to write CAD assemblies as DITTOS

CATIA V5

New export for creating a hybrid PartBody where all sheet, wire and point entities are written with solid bodies under the same Part Body set.
Export for an assembly as a single CATpart.
Export of assigned labels

DXF/DWG

3D Mesh support
DXF support for reading and writing faceted solids

FLITE3D

New spline fitting option of fixed *m**x**n* grid with local assignment
The ability to optionally export user-defined GROUP (.grp) files containing collections of related CADfix surfaces.
The ability to optionally export user-defined XONE (.xon) and ZONE (.zon) files containing the definitions of the Boundary Conditions and other Mesh Control Parameters applied to individual surfaces, collections of surfaces assembled into CADfix Zones, and lines or chains of lines.
Limit on the ratio of adjacent bands in an adaptively generated spline surface with a "Max. Growth rate" option

ICMS

Export support for wireframe curves and untrimmed surfaces. The format of the curve/surface data is the same Ferguson spline as used by FLITE3D

JTOpen

New import for JT precise BREP data
Supports parts, assemblies and attributes
Import and export available to non-JTOpen consortium members

SC03

Export of mesh
Support for face RGB colours

STEP

Support for import of faceted BREP solids

STL

Support for export of user defined source sizing controls providing greater control and improved facet quality
Extend import to read CTRIA3 elements from NASTRAN bulk data file . The import filter now includes .nas* & .bdf. The facets are filtered into regions based on material property.

Unigraphics

Reference sets

Assemblies

General improvement of performance.
New list-mode view in Assembly tree for access to instance attributes

New filtering options

Managing assemblies has been improved so that you can now select multiple components/sub-assemblies in the assembly tree. This is useful if you want to delete multiple components in one action. There is also a new component selection mechanism whereby you can select multiple components by size (either bounding box or volume) or by label (wildcard filtering applies).

New assembly/part sub-filters to main CADfix file selector. When you have a large assembly with a mixture of assembly files and part files all together it can be difficult to spot the assembly files amongst all the part files. The modification adds two new icons next to the Filetype entry field that will filter out either the assembly files or the part files. They are mutually exclusive check buttons which means you can toggle between them or switch them both off, to get back to the default list of all files. These icons only appear for systems that have different Filetypes for assembly and part files, so at the moment this is, CADDs, Inventor, SolidWorks, CATIA V5 and Pro/E.

Screen selection

Highlighting, selection and framing on specific assembly component instances. This functionality has now been integrated it into the Basic Selection mechanism. Turn on Basic Selection while the assembly tree is displayed (listed) and any object selected by making a screen pick will be highlighted in the assembly tree. Use of "Control" picks to deselect an item are also now supported.

Record/playback of edits

New record feature for items which have been deleted from the assembly tree. Choose 'Save differences' from the 'Select' filter and the list of components/sub-assemblies that have been deleted since the model was first opened will be saved to a file. File extension is .cst (i.e. CADfix Selection Template). The idea then is that after a lot of small component parts (such as nuts, bolts and washers) have been deleted from a complex assembly you don't want to spend time repeating the whole exercise the next time a new revision of the assembly is supplied. So you just load up the previous selection template and all those components previously deleted are automatically selected. User then still has the opportunity to modify this selection before invoking the deletion again.

User Interface

New image capture button replaces Print manager
New message centre tool to replace multiple info/warning pop-ups
Better spin centre selection
New CFD domains tool

New display tools

There's been requests over the years to simplify the CADfix display icons and make them more like traditional CAD systems. The drawback to doing this has been the loss of flexibility that the current display

tools gives. A small change to GUI appearance has been made with no loss of flexibility. The display tools is now a configurable toolbar. Its configurable in that you can now swap the standard "advanced" display tools for a simpler set of "basic" display tools. The "basic" set of display tools consists for:

- edge connectivity
- wireframe
- shaded
- shaded + wireframe

They are buttons like the old tools but they are different in that they do not add to the current display, they always wipe it first then display what was chosen. This gives the quick and easy option of flipping between connectivity, shaded, etc.. without having to do a wipe between each one. It also allows you to get back to a clean, initial view with one click. The ability to add these new "basic" display items to the current picture has been permitted by allowing the user to hold down the <Shift> key when pressing the basic display icon. If <Shift> is held down then no wipe is done prior to displaying the chosen action. So you can use this mechanism to build up the picture like before.

The traditional display tools are now in the "advanced" toolbar, but apart from this move they are unchanged. They can be mixed with the "basic" tools by switching between the toolbars as needed. E.g a "basic" shaded display could be drawn, then switch to the advanced tools to add "line end-points + labels", say, then back to the basic tools to add, via holding <Shift>, the connectivity display.

The colour used for the basic wireframe display is also a new configurable option under the Display settings dialog. Default is black, but if background is black then it auto-switches to dark grey.

New Application Data Tab

See below

Support for chains of lines

Under the Join Edges tool the 'Find' will reveal any chains of lines in the model and these will now be identified with a new icon in the summary at the bottom left hand of the display. The user may also open the Basic Selection tool and using the right-hand clip menu identify 'Chains' of lines by floating the mouse over the model.

Support for Zones

The Basic Selection contains an option for selecting Zones if they are present in the model. The number of Zones will be identified in the summary list at the bottom left-hand corner of the display with a new icon.

Wizard Process

Preference setting for units and an upper limit on the tolerances

The 'Max. Wizard tolerance' value can be set with an accompanying unit, e.g. "0.5 mm" or "0.1 in". Other variations on the unit specification are valid such as "0.2 mm" or "0.2 MM" or "0.2 inch" or "0.2 INCH". The value entered is checked for validity and written to your .CADfixNNN file as a space separated string. If the 'Max. Wizard tolerance' is a blank string (default) then no action is taken when the tolerances are set in the Wizard (as current behaviour). If not a blank string then any tolerance calculated in the Wizard is reset to the value specified only where it would exceed that value. The values affected by this preference setting are the repair maxTol, all prepare tolerances and the following transform tolerances: 'joinFaces,change'; 'collapse,length'; 'imprintEdges,tol' and 'imprintBodies,tol'. In the case of the prepare tolerances the wizard maxTol setting is used to limit the 'master' prepare tolerance from which all other prepare tolerances are derived. So the prepare healing tolerance may be limited to "0.1 mm", say, but the tolerances used to convert rationals to non-rationals would be set at a factor of that upper limit (should always be < wizard maxTol). Values set via a .cwc file will NOT be clipped to this upper limit.

Wizard Repair

New fix for grossly undersize surfaces

Wizard Defeaturing

Automatic Feature removal for selected Faces in a defined SET via new Defeaturing cwc options

Improved Face Joining

New Body Joining options

Wizard Prepare

- Prepare process updated on all systems for improved success rate
- More options exposed to .cwc control
- Separation of parameters for edges and surface prepare options in Wizard

Diagnostics/Interactive Repair

- New tool for creating ruled surfaces
- Selection for basic edges or combined edges
- Ability to select intermediate edges to give surfaces with internal control and sharp discontinuities

Surface Intersections

- New intersection line tool
- New split face(s) tool
- Improved surf-surf intersection GUI

Diagnostic results tree

- Now has an option to list individual entities that fail a test. Expand tree to show individual entities. Users can cycle through failed entities and apply different fixes and options to individual entities if needed

Drill-tip removal tool

- Removes drill-tips at the bottom of blind holes.

Rework of Geometry Builder GUI

- Better Create/Edit differentiation
- Vertical tabs for different build options

New face joining algorithm

- The new find function applies a set of filters to the faces to filter out those that are not joinable (such as seamless faces and drill-tips) and those which it may be undesirable to join (such as multi-loop faces, relatively large faces and multi-sided faces - all under user control). The function then builds up a set of blocking edges through which the adjacent faces will not be joined and the main criteria for placing an edge in this set is a combination of its curvature (or sag) and end-point connectivity. There is also a set of check buttons that control whether groups should be rejected if they are not four-sided. These check buttons are OFF by default.
- Added Join Faces to Neighbours as a possible repair options for different geometry problems such as Degenerate Surfaces, Degenerate Surface Corners, Self Intersecting Loops, Sharp Corner Angle, Narrow Faces, Tiny Faces...
- Performs automatic Set completion to collect neighbours for joining
- More robust narrow face/sliver face finding and fixing
- Changed layout of 'Find' and 'Fix' parameters into two separate 'hide' frames

New Body Joining

- New option to find bodies in contact with one selected body. Select required body and then "Find" touching solids. The previously "Find" would consider whole model. This is more suitable for complex assembly defeaturing and body joining

High aspect ratio face fix

- Detect faces that could be a problem for some meshing systems e.g. thin annular rings
- Automatic splitting

Global Speed Rate - New Surface Parameterisation Check

- Based on ratio of the smallest to the largest 2D parametric patch spacing in u and v (or s and t) across the full extents of a NURBS surface. If ratio is >5 then the surface is flagged as a poorly parameterised surface for meshing purposes
- Primarily for SC03 meshing but may be useful for other applications

CADfix 8.1 Data Exchange Change Notes

Defeaturing/Simplification



Splitting

Overhaul of split-face-by-shelling tool for improved robustness and performance

New Body welding

New option to build a body across a gap between two sets faces.

Includes NURBS degree raising capability

Pick position on edge close to point

A new point pick option has been added to the split by sketching tools that allow you to pick a position on an edge closest to another point. This allows you to sketch a line across a face from an existing point to a position on another edge that is opposite (or closest) to that point.

This capability has been added in other geometry build tools where appropriate. i.e. you can now create a line between two points where one or both points are defined at a position along a line closest to another point.

New sweep point/line/face geometry build option

There are three new geometry build creation methods for making a line/face/body by sweeping a point/line/face.

Application data

Facets

Improved faceted model processing tools

Improvements to the tool for selecting regions of facets (painting) for copying, deletion, extraction or replacement

Improvements to the tool for checking for faceting issues and repairing

Improved tools for facet vertex selection and manual collapse

Zones

Improved creation and editing of zones

Sources

Improved management of sources

General

Extended measure tool to pick on any position

Unicode filename support

Windows /3G support

Improved Graphics performance for large models

New frame-rate controls

The graphics has been re-organised and segregated to improve performance by utilising various culling techniques including; rotate as a wireframe, viewport culling, time limited culling, detail culling and mouse motion culling.

Change ISO view to be CAD standard

Customers have pointed out that the CADfix ISO view angle is not the same as standard CAD systems ISO view. CADfix has XY axis pointing away from user whereas standard is to have them pointing towards user, i.e. looking at "front" of object rather than "back"

New optional toolbar to ROT UP/DOWN 90.

Users wanted easier access to the rotating the view up/down or left/right by 90 degs. Useful when combined with switching to a XY view then stepping around the 6 views of an object. Current fixed views only allow 3 views to be seen. To switch from a "front" view to a "back" view hit rot-by-90 twice.

Probe by Instance number

Identify a CADfix entity by its instance number. For instance you wanted to know which face in CADfix was the 1234th face in the downstream system. To solve this the probe GUI tool will allow *(type+)instance* numbers to be typed into the entry field. An instance number must be prefixed with a "#". For the case where

the probe is closed, or you want to switch the entity type in an open probe then the instance number should be prefixed with one of the following type codes:

- P = Points
- L = Lines
- S (or X) = Shapes
- F = Faces
- B = Bodies
- N = nodes
- E = Elements

Model Tree Management

New support for zones and combined entities. Extra levels in the Model Tree hierarchy support face zones and these can be expanded to show the basic parts in each.

Mass Properties

Now supported for faceted surfaces, e.g. imported STL models

Model Management

Linux–Windows bi-directional database converter. Avoids the need to use GDx or other intermediate files to move between platforms. Executable in CADfix \bin directory:

```
\bin\cfbabel\cfbabel.exe
```

The command line usage is:

```
cfbabel.exe file1 file2
```

This will convert a CADfix database 'file1' in Windows format to 'file2' in Linux format or the reverse, if 'file1' is in Linux format

Overhaul of flexLM Licence heartbeat logic

CADfix users have proposed that the licence be lost after 15 minutes of inactivity. CADfix is currently setup to ensure a licence NEVER times out by sending regular heartbeats. A new site configuration option switches on activity timeouts, e.g:

```
*famgui.inactivityTimeouts: 1
```

If this is enabled then the GUI will stop sending heartbeats after it detects that the user has stopped working. It detects this by looking for commands/output being sent to/from the core code, or the GUI being in a busy state. Once inactive then the licence will timeout after 2 hours by default but can be pushed down to 15 minutes by setting an option in the licence server options file:

```
TIMEOUTALL 900  
(900 seconds timeout set on all features).
```

If a licence times out and then the user comes back to start working again the heartbeat will restart if the heartbeat can reacquire the licences that expired. However if it can't reacquire the licences then it falls into a heart attack mode. The user has the choice to exit or quit CADfix or try to reconnect to server. A reconnect attempt takes 40 seconds. If unsuccessful then the heart attack continues and the user can try again. Three reconnects attempts are allowed before the user must either exit or quit.