

10.0 SP1 Enhancements

Sheet Creation, Ordering, and Interaction

Sheets can now be reordered

New sheets can be named when they are inserted

User is no longer required to exit sketch mode to switch sheets

Standard View Creation Options

Standard View Creation Dialog Box new options

- Select All to use all possible automatic detailing options
- Tangent Edges
- Bend Centerlines
- Part Trails regardless of if they are on/off in the assembly

When creating standard views of Sheet Metal parts, an hourglass cursor now appears.

New Icons

New Icons

- Reproject Icon
- Reorder Sheets Icon

Exploded View Interaction

Option to show part trails with right-click menu in exploded Assembly Drawings

Locating Detail/Section View Labels

You can now drag detail view labels to where you want them; the leader line updates automatically.

You can now drag section view labels to where you want them.

Annotation Creation/Placement

Dialog Boxes

Annotation dialog boxes stay open so you can create multiple annotations at once without reopening the dialog.

- Note
- Datum
- Datum Target
- Feature Control Frame

- Surface Finish
- Weld
- BOM Callouts

Placing Annotations

- Preview is at the center of the drawing for notes by default, instead of at 0,0.
- Preview moves with mouse location
- Clicking places arrow and second click places the annotation
- Double clicking on the drawing will place the annotation; so the user does not have to always click on the Apply button.

Annotation/View/Sheet Association

Datum placement on views

- If the view is selected then the datum will be placed in that view
- If the sheet is selected then the datum will be placed in the sheet
- But if the mouse goes over a view then it will be placed on that view

Aligning Annotations to other Annotations

After placing an annotation, you can drag it and snap it relative to other annotations – each corner and centerlines of the annotation.

Dragging an annotation will snap both vertically and horizontally

- Ctrl key will override this option
- Sketch mode is not activated when an Annotation is activated
- Linear dimensions can snap to the center of the dimension lines

Copy / Paste for Annotations

Can copy/paste single or multiple items, across sheets if necessary

Will work for the following:

- Note
- Datum Target
- Feature Control Frame
- Surface Finish
- Weld

Will not work for

- Datum
- BOM Callouts
- Hole Callouts

Linking Annotations to Model Geometry

When annotations are placed they can be associated with any projected edge in a 2D drawing. The association can be made to any point on the edge.

2D Performance Options

Option to prompt user upon moving a detail/partial/section view reference to ensure they want to move the item. In some cases, accidentally moving the item on a very complex model causes long reproject times.

MultiLoop Sketches

Better tracking of faces created by multi-loop sketches. This prevents some potential ACIS problems.

API Enhancements

ExportIGES Method

Modified the ExportIGES method to use the current IGES export options from the user profile rather than the default IGES export options. To allow API users to change the export options, the new interface IADIGESOptions was created.

IADHoleFeature and IADTappedThreadInfo

Added methods for creating hole features to the IADPartFeatures class. Added properties for getting the thread info to IADHoleFeature. Created new interface IADTappedThreadInfo to contain the thread information for a hole.

Sketch Shapes can be queried

An exception was occurring because there was not yet an implementation in the API for Sketch Shapes. To allow API users to access this data, new interfaces IADSketchShapePattern and IADCompositeFigure have been created.