Contents

Chapter 1 **User Interface and Productivity 1**

Inventor Help Experience 1
Inventor Essentials Videos 2
Marking Menu 2
Custom Settings 3
Enhanced Mini-Toolbars 4
Dialog Boxes 5
Select Other - Behavior 6
Dynamic Sectioning 6
Extended Feature Names 7
High Speed Drawing Views 8
Change View Orientation 10
Rotate Drawing Views with Sketches 10
Origin Mark for Ordinate Dimension Sets 11
Wizard to Copy iLogic Designs 11
Custom User Interface Forms 12
View Display 12

Chapter 2 **Enhanced Modeling 15**

Model Repair Tools 15
2D Sketch 16
Boundary Patch Command 17
G2 Variable Radius Fillets 17

Edit Alias Freeform Body . . . 18
Mirror Fillets . . . 19
Face Draft Enhancements . . . 19
Display Failed Feature . . . 20
Boss Enhancements . . . 21
Project to 3D Sketch . . . 22
Rib Enhancements . . . 23
Sweep Along Edge . . . 24
Design View Representations in Parts . . . 25

Chapter 3 Interoperability . . . 27
AutoCAD Import . . . 27
Updated Translators . . . 28
Rhino File Translation . . . 28
BIM Exchange . . . 29
BIM Connectors . . . 30
OmniClass Table . . . 31

Chapter 4 Simulation . . . 33
Retain Joints . . . 33
Continuous Load Improvements . . . 34
Frame Analysis Probes . . . 34
Beam Model Improvements . . . 35
Customization of Beam Properties . . . 36
Sign Convention in Frame Analysis Results . . . 37
Geometry Healing and Meshing . . . 37
Guide . . . 37
In-canvas Tools . . . 38

Chapter 5 Sustainability . . . 39
Eco Materials Adviser . . . 39

Chapter 6 Tooling . . . 41
General enhancements . . . 41
Select faces to create an insert automatically . . . 42
Tooling Analysis Results in Moldflow Communicator . . . 43
Export 3D Model to Moldflow Insight . . . 44
Runner Balancing . . . 44
Mold Base Author . . . 45
Cooling Channel Check . . . 46
Cooling Channel Metric Threads . . . 47
Surface Patches with Sketch Geometry and Silhouette Curve . . . 48
Runoff Surface with Sketch Geometry and Silhouette Curve . . . 49
Patching and Runoff Surface Color . . . 49

User Mold Base 50
Edit Moldable Part 51
Transition from Part to Mold 51
Multiple Locating Rings 52
Define Workpiece Setting Enhancement 52
Retain Last Input Value 53
Gate Location X,Y,Z 54
Place Plastic Part 55
Ordinate dimensions 56

Index 57

User Interface and Productivity

Inventor Help Experience

Autodesk Inventor Wiki Help

The Wiki provides the most robust, relevant, and up to date information. In addition, the Wiki enables you to partner with Autodesk to share your knowledge with others.

- The online Wiki Help is the default help in Autodesk Inventor.
- If you are connected to the internet when you access Help in Autodesk Inventor, the Wiki Help displays.
- If you deployed Autodesk Inventor, and during the deployment you disabled the Wiki Help, it is no longer the default Help.

Basic installed Help

When you install Inventor, a subset of the help that contains customer favorites is installed by default.

- A basic Help of reference topics, tutorials for the new user, and a What's New are installed with the product.
- If you are not connected to the internet when you access Help in Inventor, and you did not download full Help, the basic installed Help displays.

Full Help download

When you are not online, you can still have access to the full Inventor Help.

- Download a full Help from autodesk.com, Data and Downloads section.

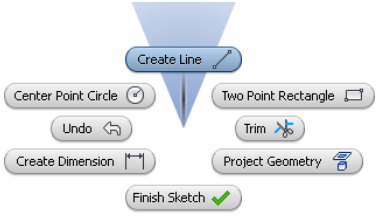
- If you are not connected to the internet when you access Help in Inventor, and you previously downloaded full Help, the full Help displays instead of the basic installed Help.

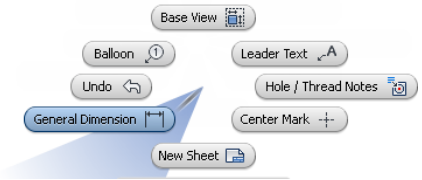
Inventor Essentials Videos

The Inventor Essentials Videos demonstrate fundamental workflows for new users. The videos include:

- Overview
- User Interface
- View Navigation
- Part Creation
- Sketching
- Assembly Creation
- Drawing Creation

Marking Menu





The marking menu replaces the right-click context menu. You can perform a selection from the marking menu in either menu mode or marking mode.

- **Menu mode**

  Right-click in the graphics window. Menu items surround the cursor and each item, or the command name and corresponding command icon display. To dismiss the marking menu, click once again in the center of the menu display. Do not press the Esc key, as this may cancel a command in progress.

- **Mark mode** (also called gesture behavior)

  Draw a mark. To enter this mode, press and hold the right mouse button, and immediately move the cursor in the direction of the desired menu item. A trail follows the cursor. Release the mouse button to select and execute the command that corresponds to the direction of movement of the cursor.

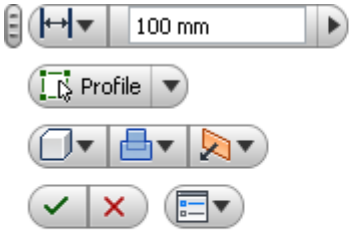
The marking menu helps to indicate next steps in a process and identify further options to facilitate workflow. Additionally, it greatly reduces mouse motion and is easily customizable to suit user preferences.

Custom Settings

Previous Inventor versions stored customized ribbon and keyboard settings in separate XML files. In Autodesk Inventor 2012, customization settings for the ribbon, keyboard, and the new marking menu are stored in a single XML file.

This new convention makes it easier and faster to restore customized settings when you install a new version of Inventor. It also facilitates sharing customized settings between different computers or with other users in your organization.

Enhanced Mini-Toolbars



Inventor 2012
Extrude Mini-toolbar

Inventor 2011
Extrude Mini-toolbar

The mini-toolbars are larger, easier to interpret, and feature greater functionality. Mini-toolbars are added for the following commands:

- Extrude
- Revolve
- Chamfer
- Fillet
- Hole
- Face Draft

There are two new mini-toolbar options:

- **Pin Mini-Toolbar Position**

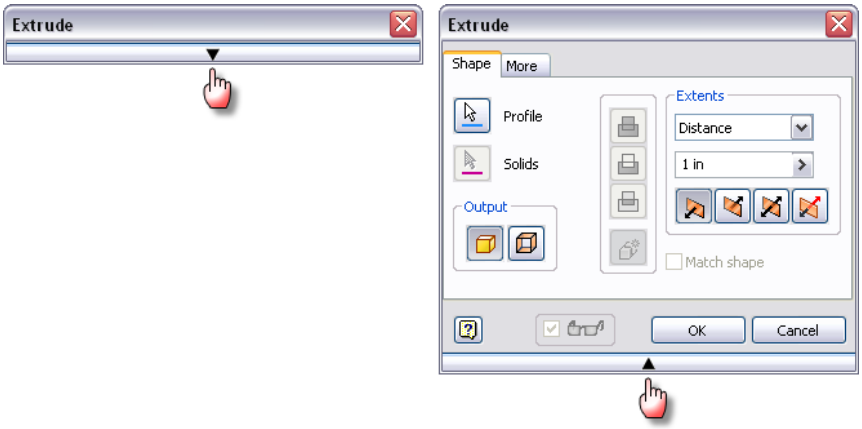
Use the grip button at the upper left of the mini-toolbar to move any mini-toolbar to a different location. You can pin the mini-toolbar so that it remains stationary in the graphics window.

- **Auto Fade**

When enabled, if you move the cursor away from the mini-toolbar, it displays in a minimized state.

When disabled, the entire mini-toolbar displays, regardless of the cursor position.

Dialog Boxes



Expand and collapse dialog boxes easily:

- To expand, click the down arrow at the top of the dialog box.
- To collapse, click the up arrow at the bottom of the dialog box.

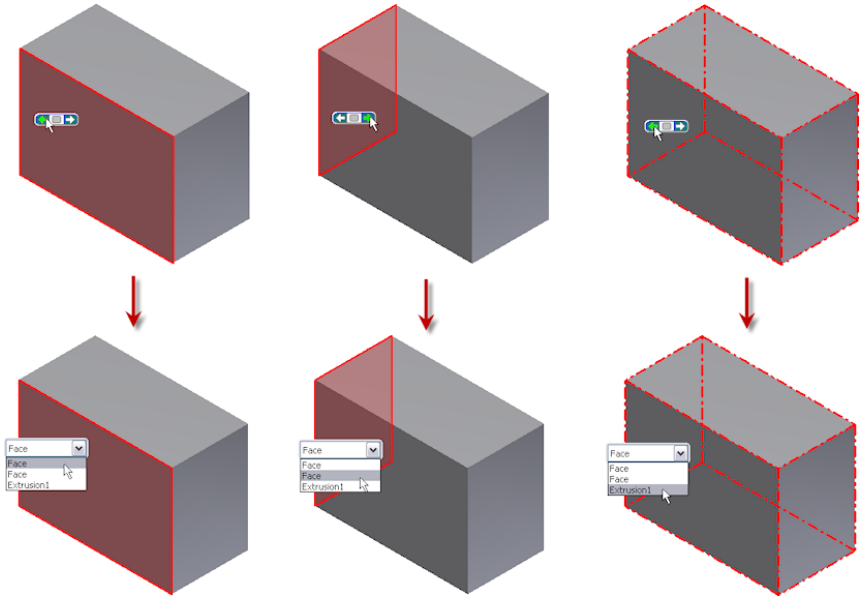
The expand and collapse feature:

- Increases available work space in the graphics window.
- Reduces mouse motion.
- Enhances your focus on the mini-toolbar command options.

Dialog boxes for commands that do not have mini-toolbars display in an expanded condition.

Dialog boxes associated with the enhanced mini-toolbars display in a collapsed condition. All of the command options are available from the enhanced mini-toolbars. To see the dialog box interface rather than the mini-toolbar, expand the collapsed dialog box.

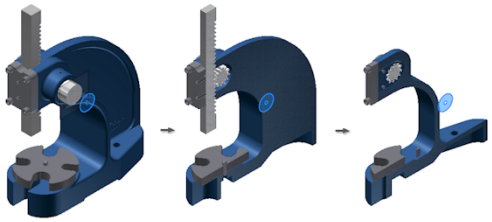
Select Other - Behavior



The Select Other procedure in previous Inventor versions required you to hover your cursor over an object, and then click the left or right arrows to toggle between adjacent faces, edges, and features.

Hovering your cursor over an object in Autodesk Inventor 2012 presents you with a drop-down menu listing all available selection options. You can eliminate guesswork and speed the selection process by making your selection directly from the list.

Dynamic Sectioning



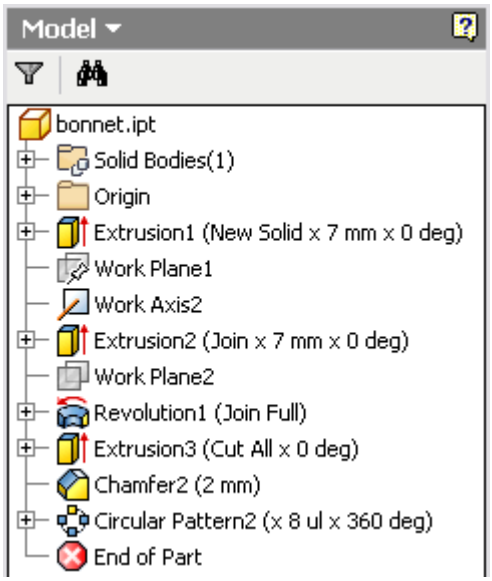
You can offset section planes in a part or assembly section view.

Right-click, and select Virtual Movement ➤ Section Plane 1 (or Section Plane 2). Then:

- Drag the section plane to the desired position.
- Specify on offset in the Offset dialog box.
- Place the cursor on the Offset dialog box, and rotate the mouse wheel to move the section plane.

  TIP To adjust the mouse wheel scroll, right-click and select Virtual Movement ➤ Scroll Step Size.

Extended Feature Names



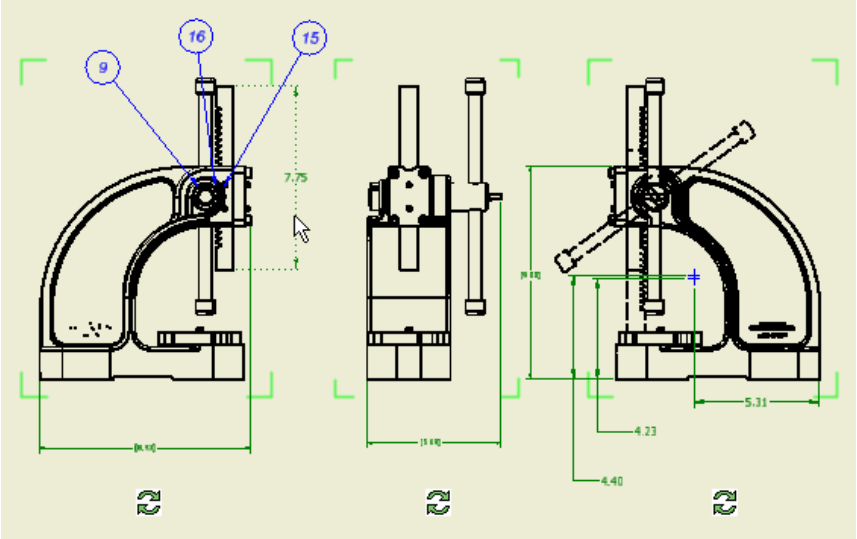
In the Model browser, the names of part features can be extended with information about feature parameters. Extended feature names enable better identification of existing part features.

To display extended feature names, on the Browser Filters menu

, select Show Extended Names. Extended feature names are displayed in the Part, Sheet Metal Part, Assembly Modeling View, and Drawing environments.

NOTE You cannot edit the content or format of the feature name extensions.

High Speed Drawing Views



You can review the drawing or create drawing annotations before precise calculations of drawing views finish. Precise drawing views are calculated in the background while you work with draft previews (raster views).

Raster views are marked by green corner glyphs in the graphic window, and by a special icon in the browser. If you place the cursor on a raster view icon, a tooltip showing the progress of precise calculation displays.

NOTE You control the use of raster views on the Drawing tab of the Application Options dialog box. Select the Enable Background Updates option to display raster views. Clear the Enable Background Updates box to skip the display of raster views and wait until all views are precisely calculated.

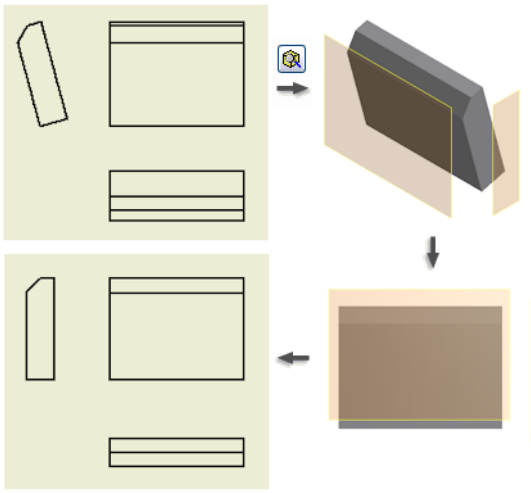
Features not available for raster views

Automated Center-lines	Cannot be created for raster views.
Auto Balloon	The command cannot be used for raster views.
Model Features	Cannot be selected as edges in raster views.

Features working differently for raster views

Tangent Model Edges	Are always shown in raster views. Their properties or visibility cannot be edited.
Interference Edges	Are never shown in raster views.
Reference Parts	Their geometry can be incomplete in raster views.
Hole Tables	View and Feature options are not available for raster views.
Thread Annotations	Thread features are not displayed in raster views. New or existing thread notes are attached to thread features after views turn precise.
Print	When a drawing includes raster views, you can print the current views, wait until all views are precise, or cancel the Print. **TIP** We do not recommend printing of raster views, because geometry on printed raster views can be different from the precise views.
Export to AutoCAD DWG, DWF, DXF, or PDF	Cannot be finished for a drawing with raster views. If a view calculation is in progress, a progress bar is displayed. You can wait until the calculation finishes or cancel the Export.
Save and Close	Drawings containing raster views can be saved and closed. On reopen, raster views are automatically recalculated as precise.

Change View Orientation



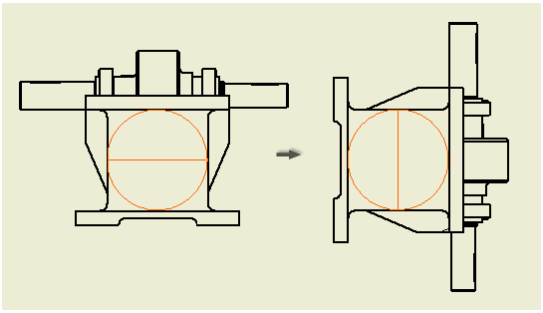
You can change the orientation of existing drawing views.

Double-click a view, and in the Drawing View dialog box, click Change View Orientation

. Then, in the Custom View window, specify a new view plane.

When you change the orientation of a base view, all dependent child views inherit the new orientation.

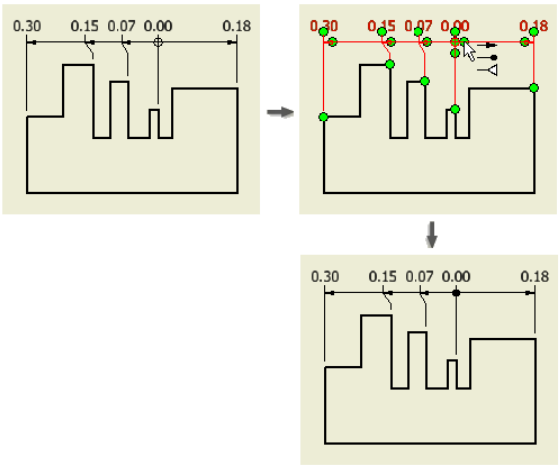
Rotate Drawing Views with Sketches



The Rotate View command is available for drawing views that include sketches.

Only sketches that are constrained to projected model geometry rotate correctly with the drawing view.

Origin Mark for Ordinate Dimension Sets



You can override the style setting and change the origin mark for ordinate dimension sets. Double-click the grip point next to the origin mark. Then select a new origin mark from the list.

To change the default origin mark for ordinate dimension sets, edit the dimension style. When you change the origin mark in the style, the existing ordinate dimension sets with By Style origin mark display the new mark.

Wizard to Copy iLogic Designs

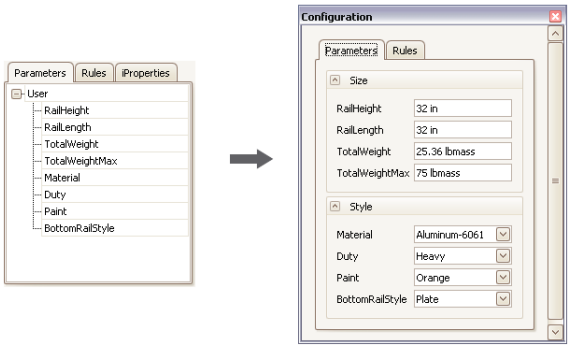
Provides a wizard to copy an Inventor iLogic template project. Use an Inventor iLogic project as a template to produce design variations. Embed design copies as subassemblies in larger designs, or as copies of a unique and complete product definition.

Features include:

- Selects linked documents automatically.
- Copy files to a new or existing project destination.

- Copy Inventor and non-Inventor files.
- Rename file copies by defining a naming prefix, suffix, or both, that can be applied globally to the copy set.
- Set the Part Number (iProperty) of the copy destination file the same as the copy file name.
- Remove iLogic rules from Inventor documents included in the copied set.
- Use even if the documents do not involve iLogic rules.

Custom User Interface Forms



Design your own user interface using drag and drop methods in a fraction of the time it took to use programming methods. Create a custom form to drive the model.

You can use the controls in the forms to drive changes to an existing design. Either save form definitions with a design document, or store them for use across multiple documents.

Create and connect the custom user interface to the parameters, properties, and rules of an Inventor part, assembly, or drawing document.

View Display

Ray tracing is added as a view setting option for the Realistic (default: enabled) and Monochrome (default: disabled) visual styles. If you select Realistic, the display refreshes using ray tracing.

You can interrupt the process with the Orbit command, after which the display refreshes with ray tracing again.

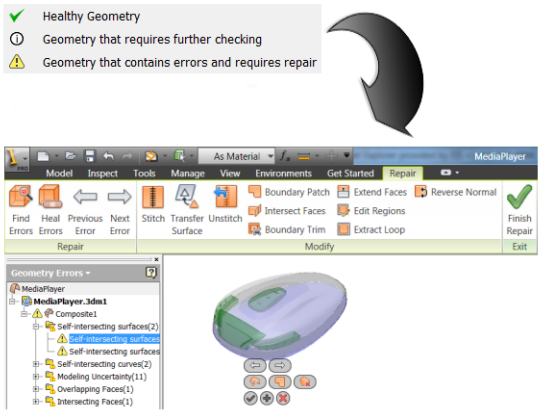
Methods to disable ray tracing:

- In the View tab ➤ Appearance panel ➤ click Ray Tracing to enable or disable it.
- Activate a command that disables it.
- Click the X in the progress bar, when displayed.
- In the Application Options or Document Settings, change the Display Appearance settings.

Enhanced Modeling

2

Model Repair Tools



Inspect and repair imported data in a repair environment for imported files. Then the imported data is reliable in downstream modeling operations.

Methods to create a repair body:

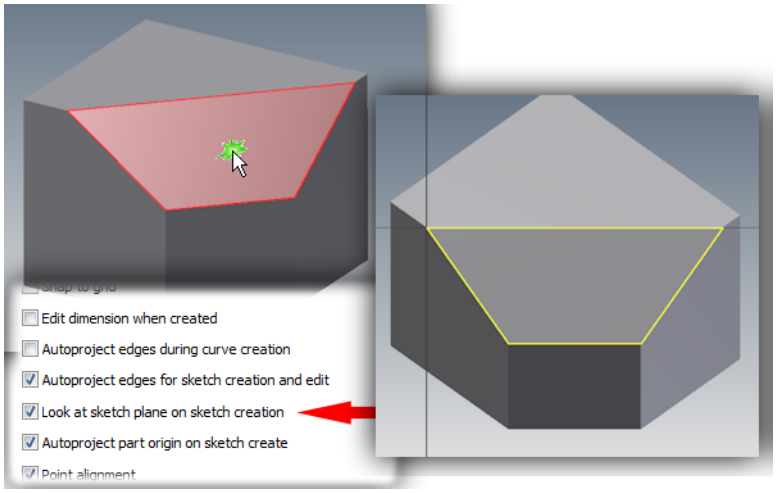
- Import the file into the repair environment using selected translators.
- In the Surface panel, click Repair Bodies and then select a base body.
- In the Modify panel, click Copy Object to create a base body and then click Repair Bodies in the surface panel.

Methods to enter the repair environment:

- In the browser, double-click the repair body node.
- In the browser, right-click the repair body node and then select Repair Bodies.

Use the Heal Errors command to remove errors with a specified tolerance value. Click an error to find it in the graphics window and enable the repair mini-toolbar.

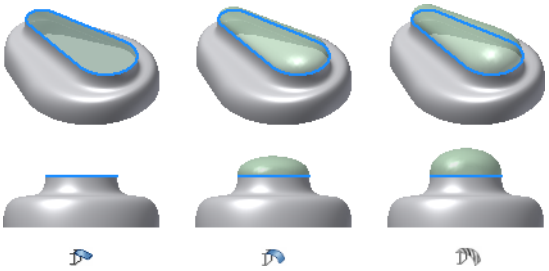
2D Sketch



The View default behavior for 2D sketches is changed. When you start a sketch on a component face, the view orients so that the sketch plane normal points at the display. The sketch plane is parallel to the display, and is ready for sketching.

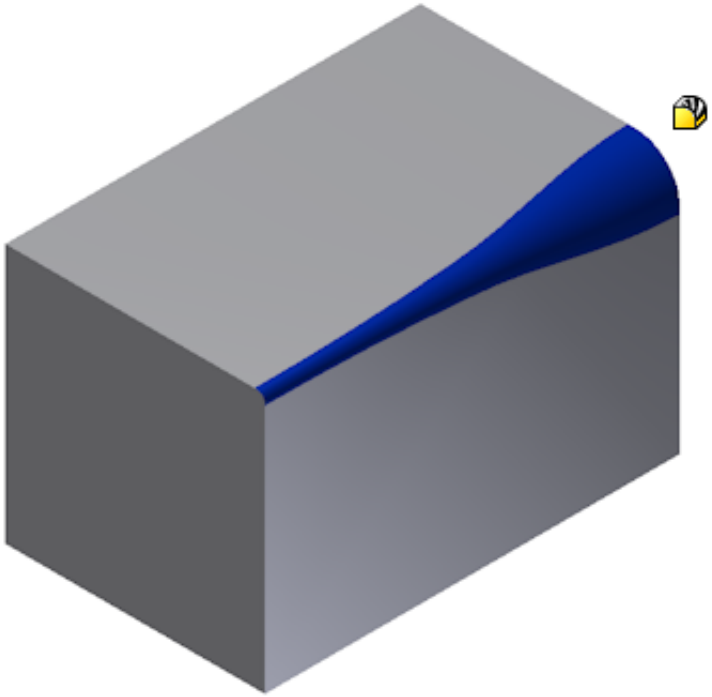
You can change the default behavior in the Application Options ➤ Sketch tab ➤ **Look at sketch plane on sketch creation.**

Boundary Patch Command



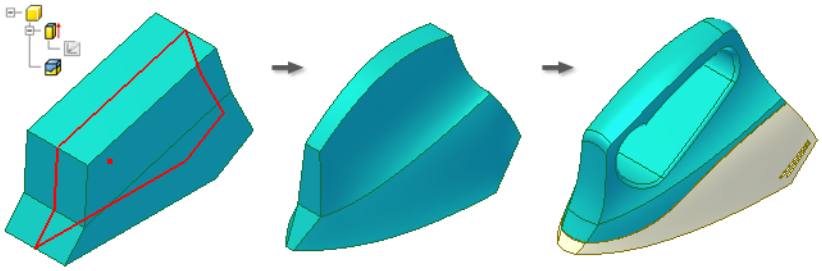
The option to generate a G2 (smooth) boundary patch is added to the part modeling and repair environment. Use a G2 transition to eliminate visible seams between surfaces.

G2 Variable Radius Fillets



Variable radius fillets support G2 control. Use a G2 transition to eliminate visible seams between faces.

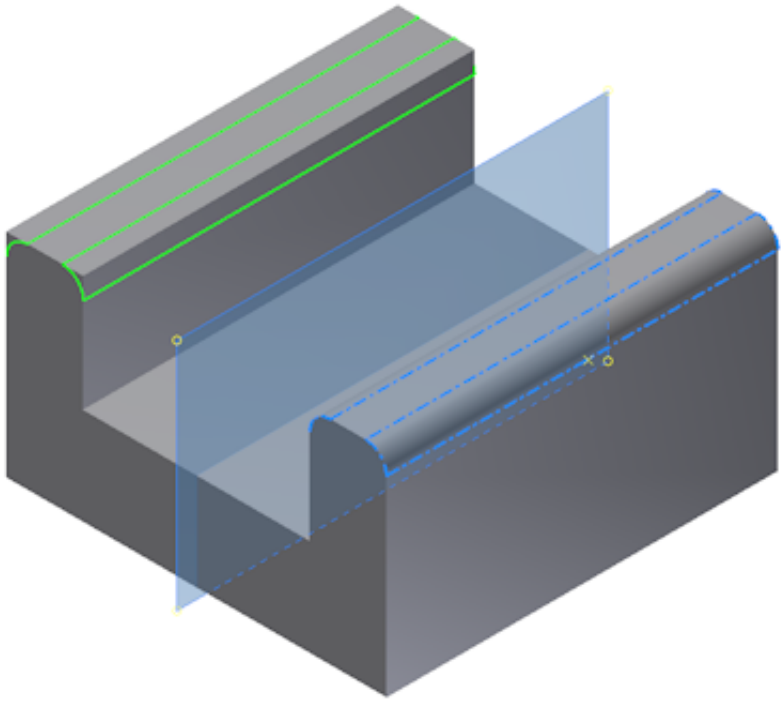
Edit Alias Freeform Body



Launch Inventor Fusion from within Inventor using Edit Form or Edit Copy of Form. The Edit Form command creates an Alias Freeform body which is then loaded into Inventor Fusion. You can modify the body using any of the Fusion commands then return the geometry to Inventor.

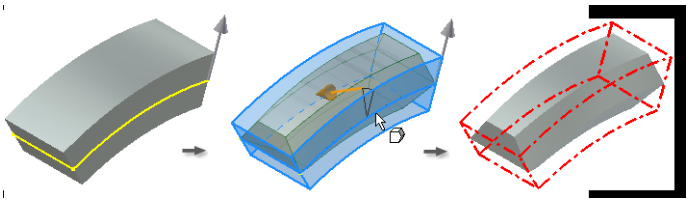
Inventor Fusion contains many of the commands previously available in Alias Design for Inventor. These features provide technology to reshape solid body edges and patches, transforming geometric forms to more organic ones.

Mirror Fillets



Fillet features can be mirrored without including the parent feature. You can create symmetric models using fewer features.

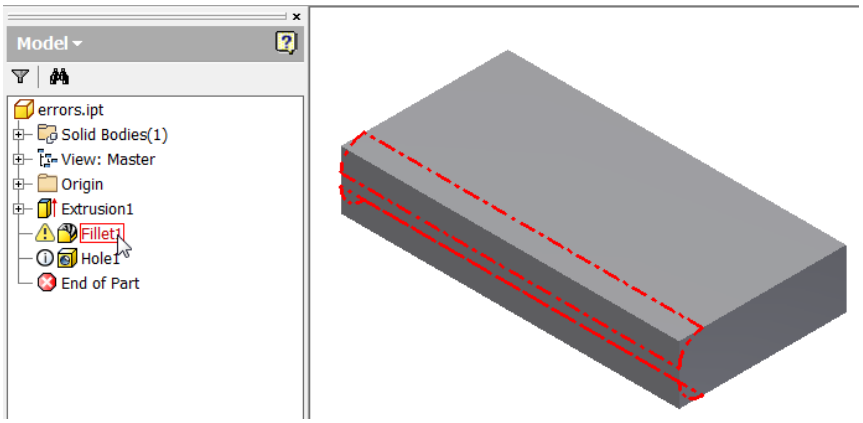
Face Draft Enhancements



Controls are enhanced in the Draft command for adding draft to a model. These controls speed the creation of draft features and make it possible to create geometry that previously required multiple features.

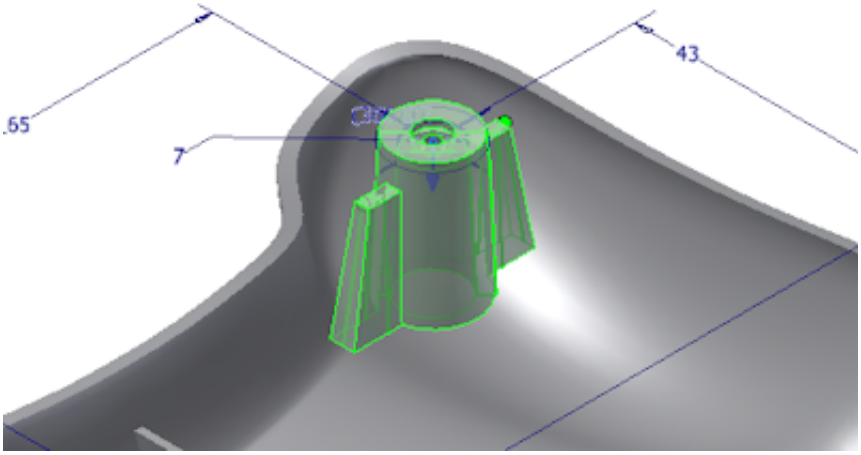
- Create a draft feature using a parting line (2D or 3D sketch) to position the start of the draft.
- Use separate angle values above and below the parting line or fixed plane.
- With Automatic Face Chain enabled, faces tangent to the selected face are included in the draft.
- Automatic Blending is available for adjacent faces that blend with a fillet or other feature. Turn on Automatic Blending to maintain blended geometry.
- Direct Manipulation controls are included with the Draft command.

Display Failed Feature



You can locate failed features easily for problem solving. In the browser, pause the cursor over a failed feature to highlight it in the graphics window in its last successful state.

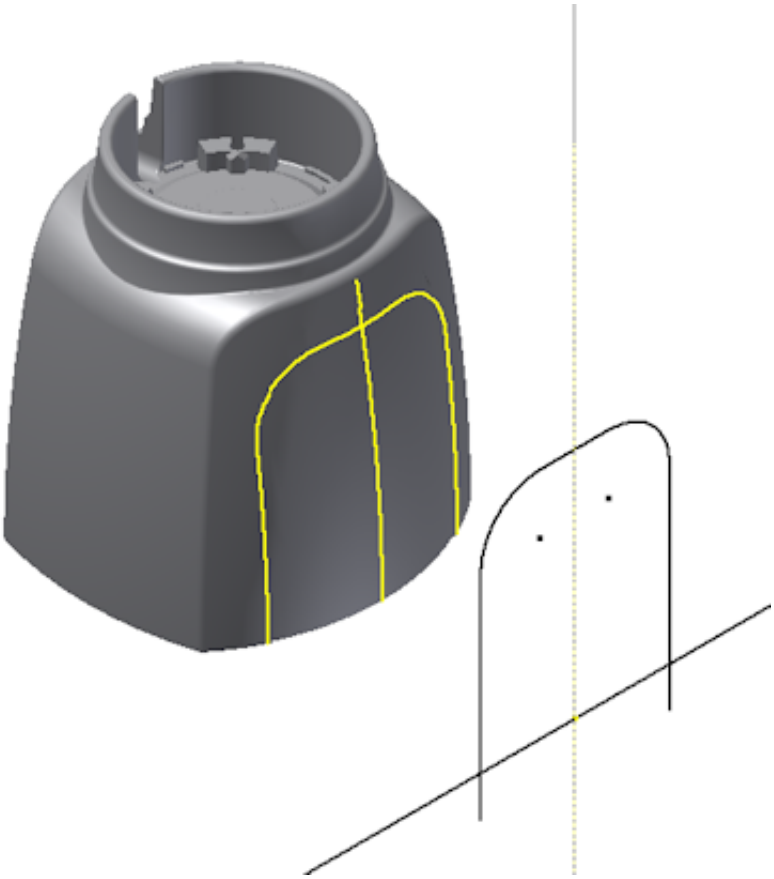
Boss Enhancements



With new controls, you can create geometry in a single boss feature that would require multilple features previously.

- A Boss feature is placed at a point centered on the mating face of the head and the thread. To change the position of a Boss placement, in the Offset From Sketch field, enter a value.
- The Angle to Fill field controls the position of ribs on the boss feature.
- You can create designs that do not require ribs to go 360 degrees around the boss.

Project to 3D Sketch

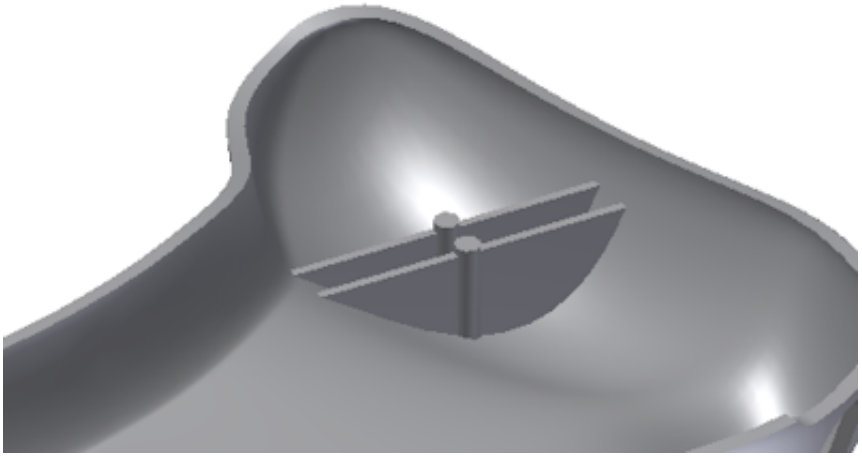


While working in a 2D sketch, you can preview 3D sketch geometry. The Project to 3D Sketch command projects geometry from the active 2D sketch and onto selected faces to create a 3D sketch.

With the command active while you modify a 2D sketch, the new geometry displays in a preview. After you finish the 2D sketch, the 3D sketch is created.

The projected geometry is associative and updates with the parent geometry. You can disassociate the projected geometry by editing the 3D sketch and using Break Link.

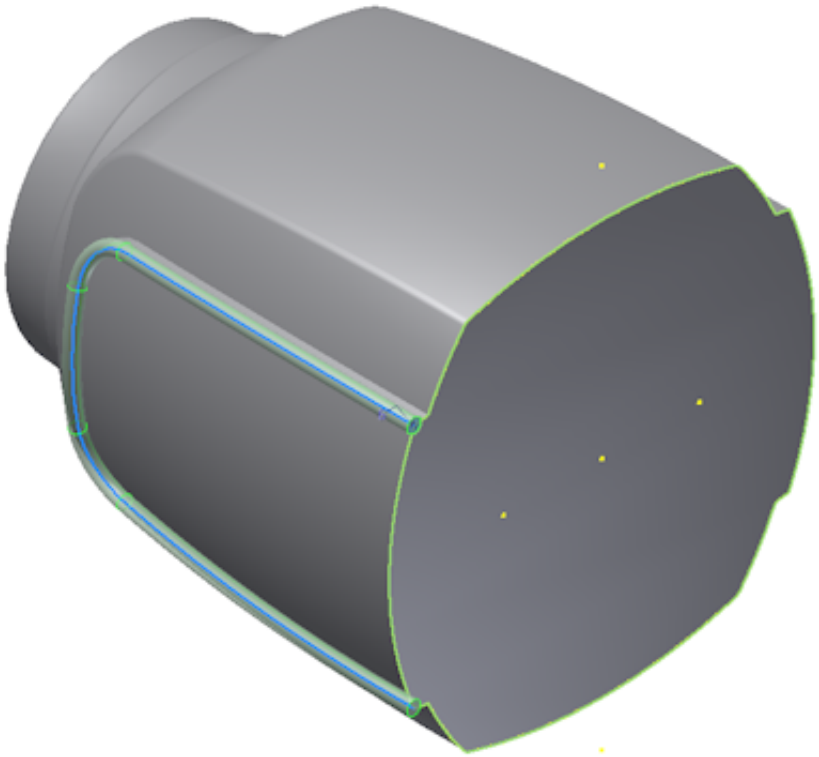
Rib Enhancements



The Rib command dialog box is enhanced with better organization and additional functionality. With the new functionality, you can create geometry that was not possible previously in a single feature.

- When creating webs (feature is extruded normal to the sketch plane), Draft and Boss tabs are available.
- An option is available to specify the thickness at the sketch or at the root of the rib feature.
- While you create rib features, a realistic preview displays.

Sweep Along Edge



Edges of geometry are valid selections for the sweep path. When you create the sweep feature, the edges project onto a new 3D sketch.

Using edges as a sweep path is not available for assembly sweep features.

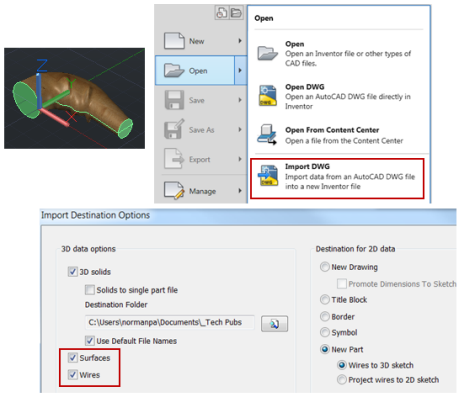
Design View Representations in Parts

In part files, design view representations store information related to the part view, such as part color, work feature visibility, and camera position.

The browser provides access to design view representations.

Interoperability 3

AutoCAD Import



Several new options improve interoperability with AutoCAD.

- On the Open flyout, select Import DWG to bypass the Options setting.
- In the file import wizard, the final dialog box contains options to import AutoCAD Surfaces and Wires. To enable the Surface and Wire selection, select New Part.
- Materials applied in the AutoCAD file persist in the Inventor file after translation.

TIP To view the material in Inventor, enable the Realistic Visual Style.

Updated Translators

The following translators support newer versions of the file format:

CATIA V5

- Import: R6 - R20
- Export: R10 - R19

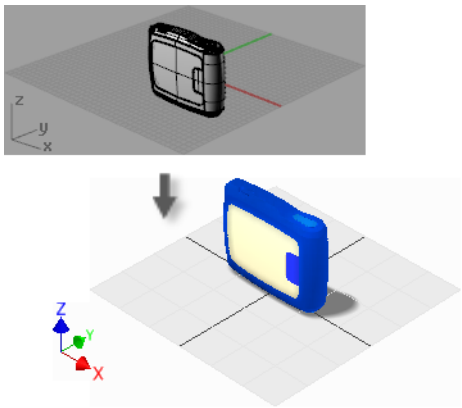
Parasolid

- Import: up to version 23.0
- Export: versions 9.0 - 23.0

NX

- Import: versions 3 - 7.5

Rhino File Translation



Rhino (*.3dm) files can be imported for use in Autodesk Inventor. The import operation does not maintain associativity with the original file. After the import operation:

- Changes to the original file do not affect the imported part.
- Changes to the imported part do not affect the original file.
- You can change the model as if it was created in Autodesk Inventor.

TIP Use the Repair environment to heal models and fix surfacing errors.

BIM Exchange

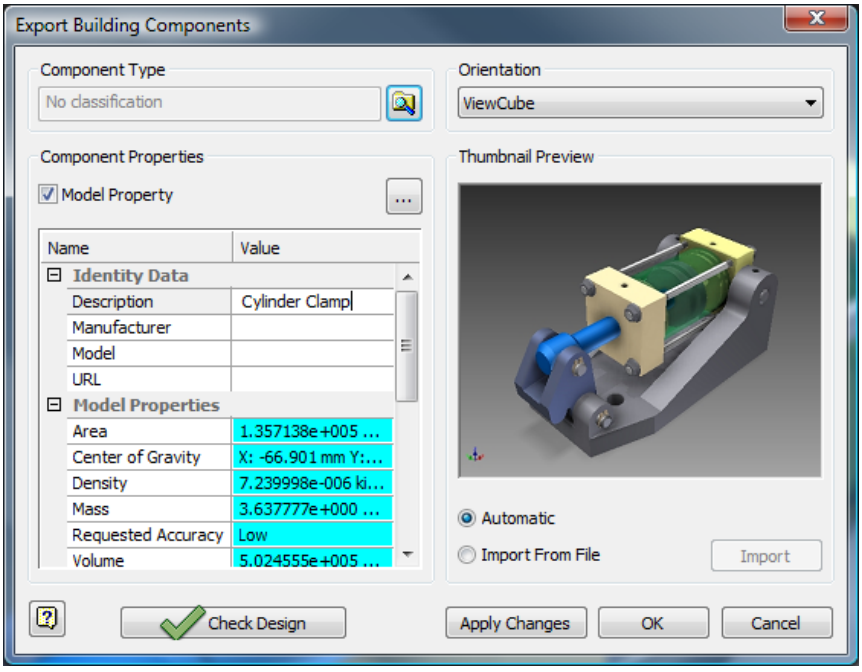
General Improvements

Several integration improvements are provided for BIM Exchange, these include:

- iPart and iAssembly table access allows you to change between members
- Export Building Components workflow improvements
- Connector access and workflow improvements

The various changes are outlined in the following.

Export Building Components



- Before you export the component, provide all inputs and selections, and then click Apply Changes.

- To exclude model properties, clear the Model Property check box. When Model Property is selected, model properties are exported with the component.
- The model orientation options include ViewCube. The ViewCube "views" generate component views when exporting. When the model is opened in Revit, the ViewCube positions sync with the Revit ViewCube.
- To redefine the ViewCube origin point, use the triad. Select a component vertex or work point as input for the origin point.
- Thumbnail images are the isometric view of each orientation option. Each orientation option has an associated image. The images reflect what you see in the display. Use View settings for shadows, ground reflections, and lighting to produce a high quality thumbnail image.

BIM Connectors

Several improvements have been made to BIM connectors.

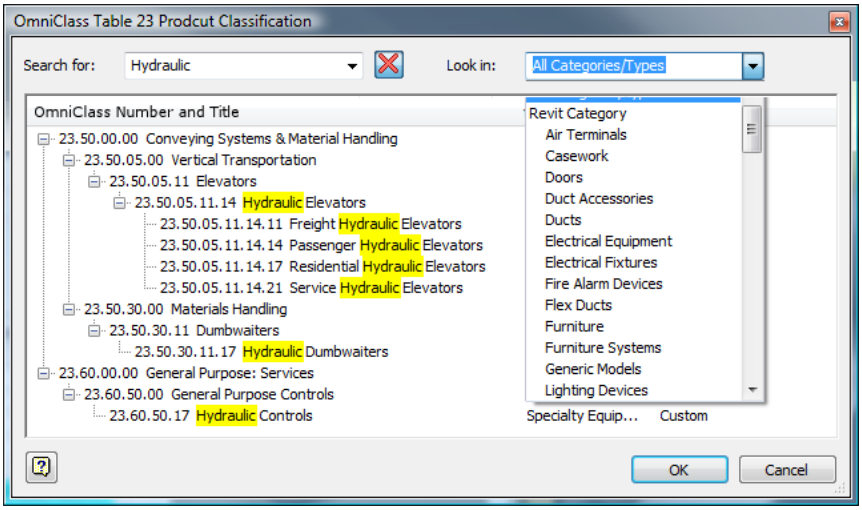
The connector improvements include:

- Parameters use Inventor unit formatting.
- Parameter values can be included in the parameters table by checking the box in the f_x column.
- For numeric parameters, the Inventor fly out menu is added providing access to Measure, Show Dimension, List parameters, and Most Recently Used (MRU) options.
- Parameters that are identified for export in the model parameter table are included in the export building component process.
- Inventor size parameters are associated with the Revit Length type.
- Connector parameters are included in the Design Check and display an icon and are colored red when invalid. If you use Export Building Components without doing the Design Check, it is run automatically and reports any invalid connector parameters. The connector browser icon updates to note the invalid connector.
- Suppressed connector nodes and text display their suppressed state.
- Exported connector parameters are accessible in the iPart Author and can be used as keys in the iPart or iAssembly.
- Promote Component Connectors enables exposing the sub-component connectors in the parent assembly. Connector links are not promoted, but

can be added. Promote Component Connectors is accessible from the connector folder browser node context menu.

- After promoting connectors, you can exclude individual connectors by unselecting Include in the context menu.
- From a promoted connector you can locate and select the Source Component via the context menu. The source component is selected and highlighted.
- Promoted connectors are included in the Design Check.

OmniClass Table



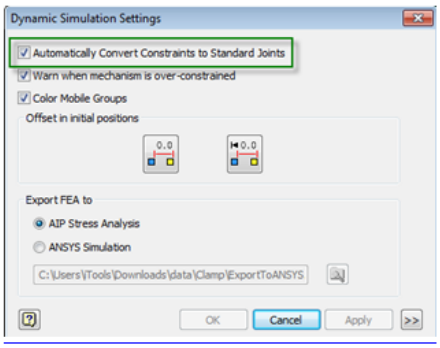
The OmniClass Table dialog box provides an improved means of locating the Revit class for your BIM content.

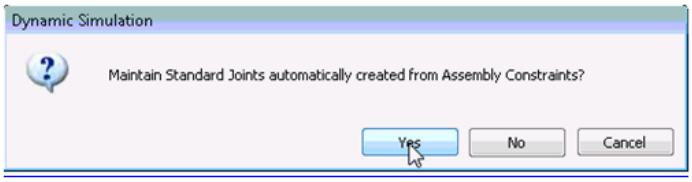
- Revit and AME MvPart types are synchronized.
- Look in option is available to refine category filtering.
- Table context menu options control column display.
- No classification option is available for your content.

Simulation

4

Retain Joints

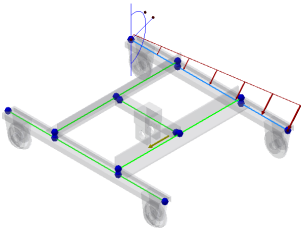




When joints are built automatically from assembly constraints, you can retain them in the model when the translator is turned off.

You have the opportunity to both customize the standard joints built from constraints and author new standard joints, without starting from scratch.

Continuous Load Improvements



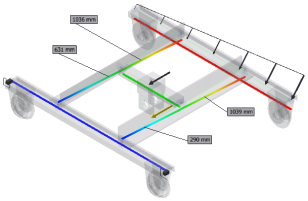
Enhancements are added to help you define the start and end magnitudes of the continuous load.

Placement options are expanded with relative and absolute definition of offset.

In the Frame Analysis environment:

- Define the end magnitude as well as the start magnitude of the continuous load.
- Checked Custom size option enables you to specify the offset and length of the load on the selected beam.
- Define offset using the absolute or relative values.

Frame Analysis Probes



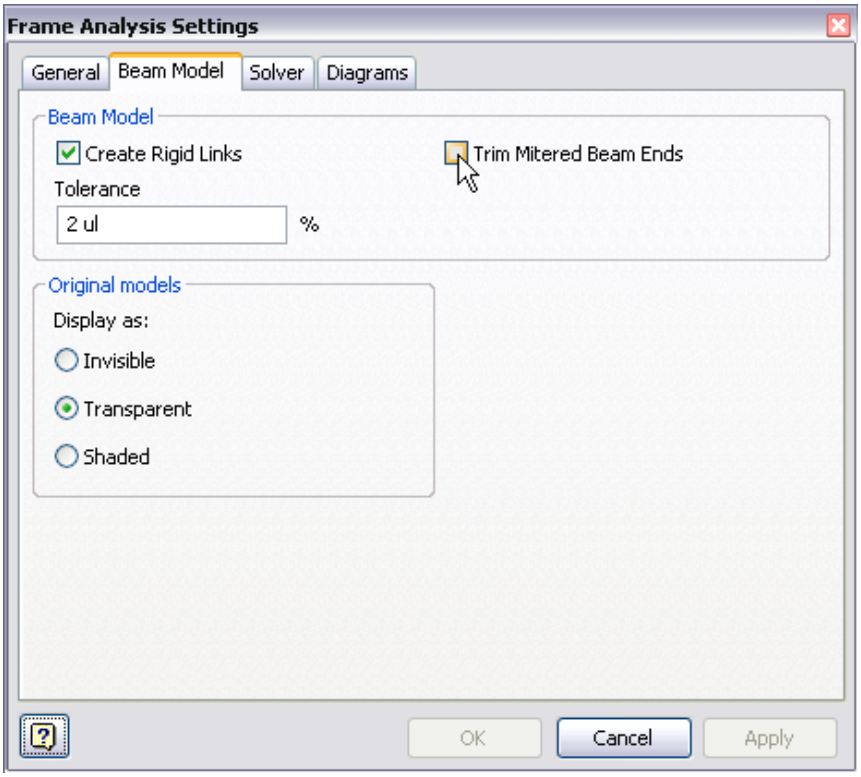
New Probe and Probe Labels commands enlarge the list of tools for viewing simulation results. The commands help you to:

- Create probes for selected simulation results.
- Control the visibility of probe label in the graphics window.

You can create one or more probes when any simulation result is active, and then switch between results to see values. Run the simulation, select Probe, and click a node or any position on the beam in your model.

Created probe annotations are displayed when any result is active. Click Probe Labels to toggle the probe visibility.

Beam Model Improvements



In the Frame Analysis Settings dialog box, Beam Model tab, two options are added:

Create Rigid Links

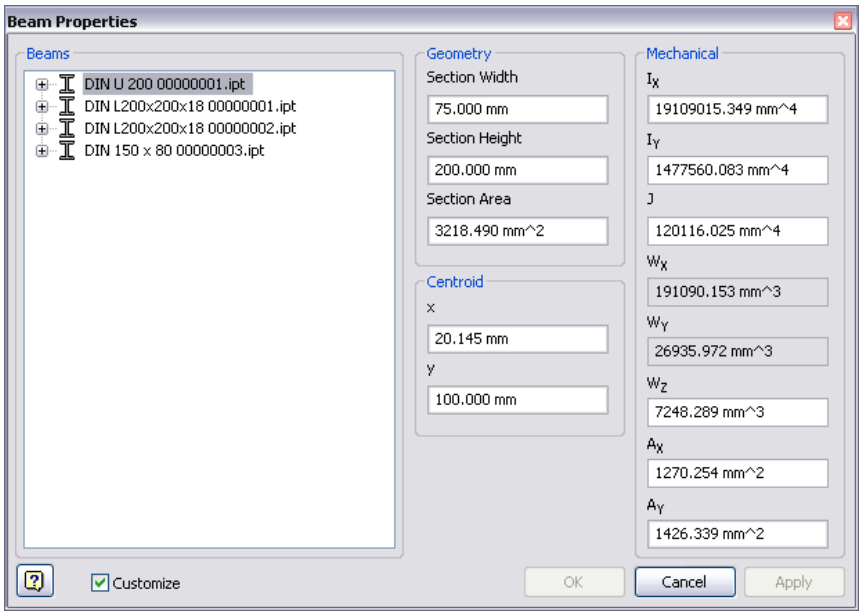
- When selected, increases the flexibility to adjust the beam model during automatic conversion.
- When the check box is cleared, rigid links are not automatically created during model conversion. Instead, you can create rigid links manually according to your needs and preferences.

Trim Mitered Beam Ends

- When selected, the overlaps are trimmed during automatic model conversion.
- Simplifies generated model and improves the accuracy of the simulation results.

For More Information

Customization of Beam Properties



You can edit an increased number of beam mechanical properties, including two new properties.

- Reduced Shear Stress (Ax)
- Reduced Shear Area (Ay

At the bottom of the Beam Properties dialog box, check Customize, and then enter your values. It allows you to fully define, or control the section properties.

For More Information

Sign Convention in Frame Analysis Results

Stress results were unified to use the same sign convention as the Stress Analysis results. The interpretation of Frame Analysis results is now consistent with Stress Analysis results.

Geometry Healing and Meshing

Enhancements in the geometry healing and meshing operations for stress analysis mean increased success in analyzing complex designs.

In consumer products and some other industries, product designs involve many surfaces and tight tolerances. You can simulate the behavior of such designs before you build a physical prototype.

Guide

The improved content and interface in the Stress Analysis Guide help you find the answers you need, whether your skills are novice or intermediate.

In-canvas Tools

New Marking Menus are present in all Simulation environments. These menus provide immediate access in the canvas to the most commonly used commands.

Sustainability 5

Eco Materials Adviser

The Eco Materials Adviser tool enables you to analyze the environmental impact of your product early in the design process. The analysis provides rapid estimates of key environmental indicators such as energy usage, CO2 footprint, water usage, compliance with European Restriction on Hazardous Substances (RoHS), end of life, and material cost, that are influenced by your material and production process choices. By browsing and selecting from the hosted materials database from Granta Design, you can then assign and evaluate alternative materials, to help minimize overall environmental impact.

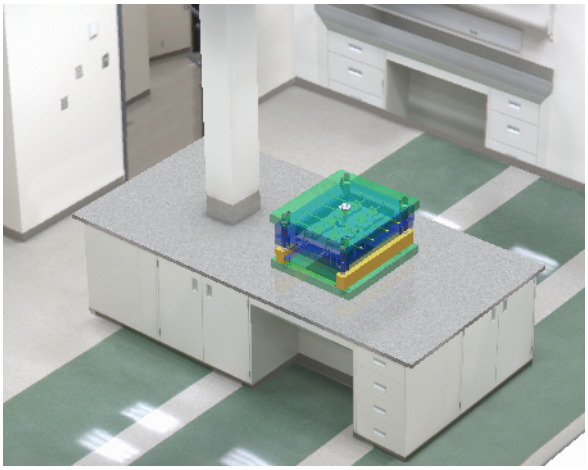
The base version of Eco Materials Adviser which is included in Inventor can analyze assemblies of up to 20 parts, and uses a starter database of typical materials. To analyze an unlimited number of parts and to get access to a more comprehensive materials database, you can upgrade to the full version of Eco Materials Adviser that is offered by Granta Design. Click *here* to obtain more information.

The tool is accessible from the Environments tab in parts and assemblies.

Eco Materials Adviser was developed as a joint partnership between Autodesk and *Granta Design.*

Tooling 6

General enhancements

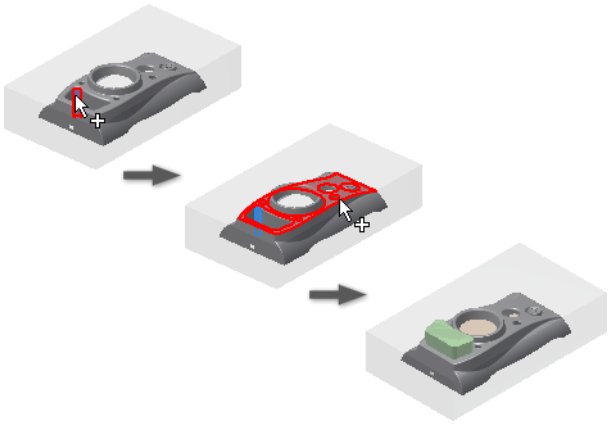


The following productivity improvements are added to Inventor tooling:

- Graphics processor support is added for Moldflow analysis.
- You can place the locating ring before you place the sprue bushing.
- Expanded LKM moldbase content is added to the library.
- A preview graphic is added to the Adjust Orientation command.
- You can stop the automatic surface patch or runoff surface operation at any time during creation.
- Edit coordinate system is added to the Ejector dialog box to redefine the origin and XY orientation.

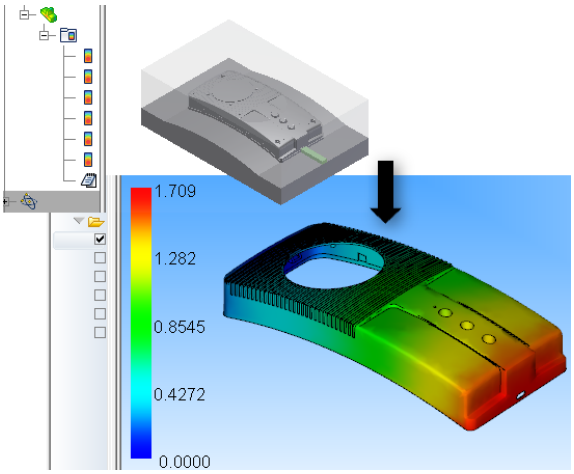
- Error messages are expanded and enhanced in many areas to provide a clear understanding of the error.
- You can specify the vendor and category for library components such as ejectors and sliders before you query the content.
- The Combine Cores and Cavities dialog box contains a preview enable/disable check box to speed dialog box display time.
- To reduce install time, only a subset of the Inventor Mold Metric desktop content library is included on the installation disk. Download all of the available content from: *www.autodesk.com\inventor-tooling-mold-libraries*

Select faces to create an insert automatically



You can use the By Shape method to select all the faces required to create an insert automatically. Use the Face Set Tool selectors to select the Seed and Boundary faces. If the boundary faces are adjacent to the seed face, check Automatic profile loops before selecting to chain the faces.

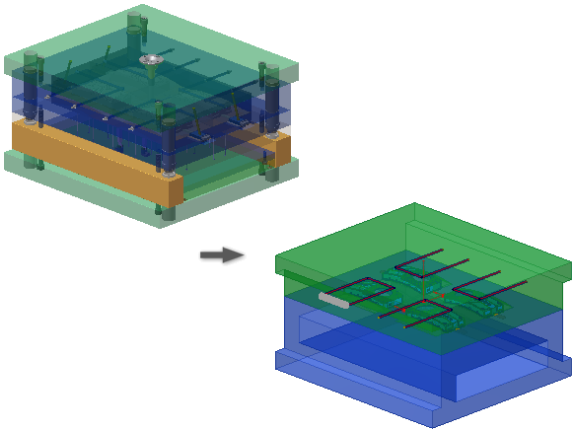
Tooling Analysis Results in Moldflow Communicator



You can export analysis results to a Moldflow Communicator file.

With Moldflow Communicator, a free viewer, others on the design team can visualize, quantify, and compare simulation results. Sharing the data improves collaboration with the extended design team.

Export 3D Model to Moldflow Insight

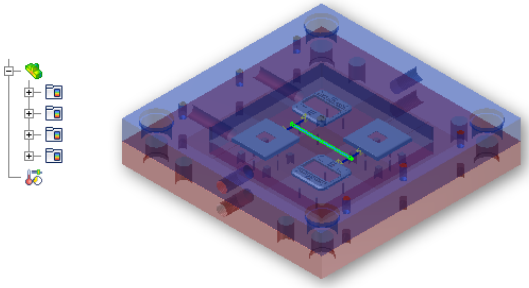


You can export the 3D mold base information to Moldflow Insight for cooling analysis.

- The mold block is exported by default.
- Specify the inserts, cooling system and components to export.
- If necessary, you can add hose definitions in the dialog box before export.
- The export process removes unnecessary hole features from the core/cavity solid.

Open the file with Moldflow Insight to validate and optimize plastic parts, injection molds, and the injection molding process.

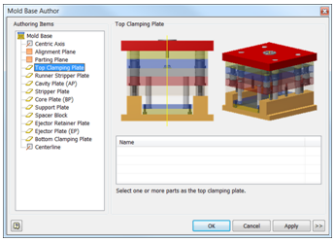
Runner Balancing

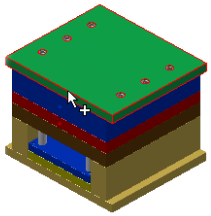


If Moldflow Adviser is installed on the system, you can use the Runner Balance command to perform a runner balance analysis on a multi-cavity or family mold design.

After the analysis completes, apply the results to balance the flow into each cavity and achieve uniform part quality.

Mold Base Author



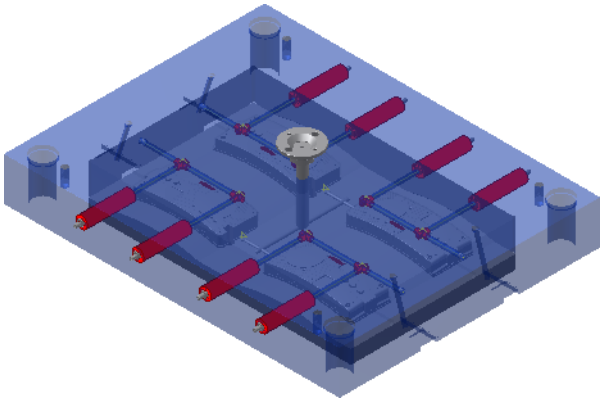


The command Mold Base Author identifies the members of an assembly as mold base components, and specifies the alignment options and mold base structure. This command is available in two locations:

- In an assembly, choose the Manage tab, Author Panel.
- In a mold design, choose the Mold Assembly tab, Author panel.

TIP Use the Mold Base Author command before you place the mold base in a mold design.

Cooling Channel Check



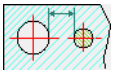
Use the Cooling Channel Check command to analyze the distance between cooling channels and other elements of the mold base.

Specify the Minimum Distance and then check for clearance, spacing, interference, and other violations. Right-click an error in the report and use Find in Window to locate the error in the graphics window.

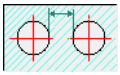
The Cooling Channel Check supports three types of checking:

- Safety Check

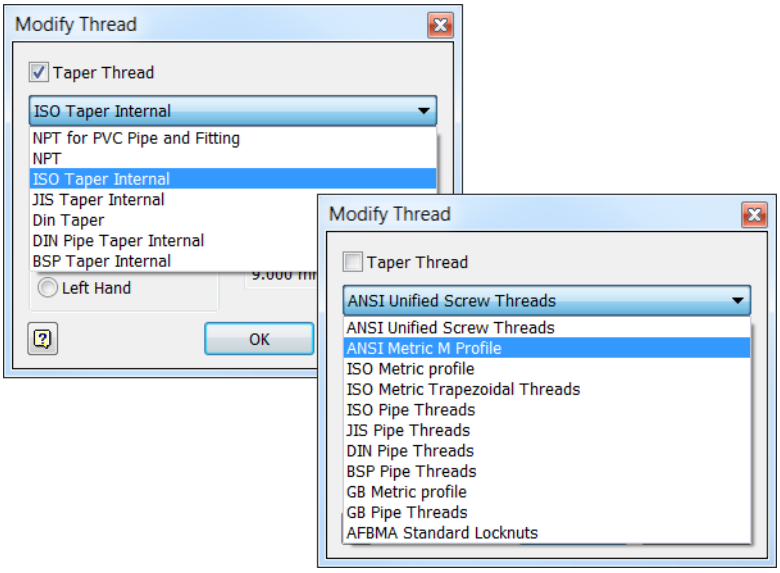
- Interference Check



- Cooling System Check



Cooling Channel Metric Threads



You can use metric threads in the cooling channel.

- To list the tapered thread options, select Taper Thread.
- To list straight thread options, clear the check box for Taper Thread.

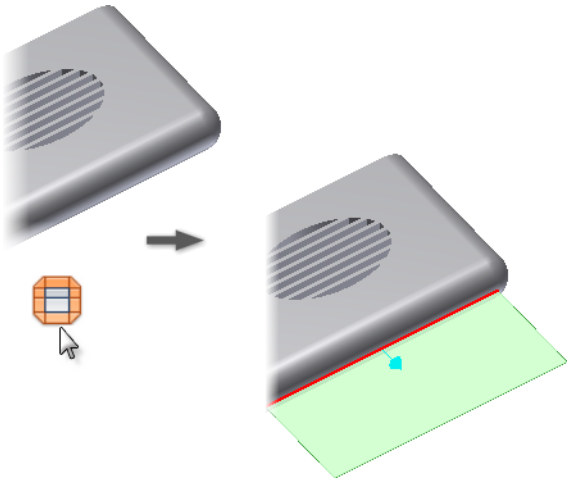
TIP To modify the available sizes and set the default selections, edit the Thread.xls file.

Surface Patches with Sketch Geometry and Silhouette Curve

You can use any combination of 2D or 3D sketch geometry and silhouette curves to create surface patches.

To generate a silhouette curve, start the patching surface command and select a face where no edge exists. The curve is used with 2D or 3D sketch geometry to create a surface patch.

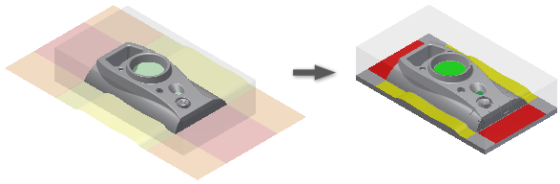
Runoff Surface with Sketch Geometry and Silhouette Curve



You can use any combination of 2D or 3D sketch geometry and silhouette curves to create runoff surfaces.

To generate a silhouette curve, start the runoff surface command and select a face where no edge exists. The curve is used with 2D or 3D sketch geometry to create a runoff surface.

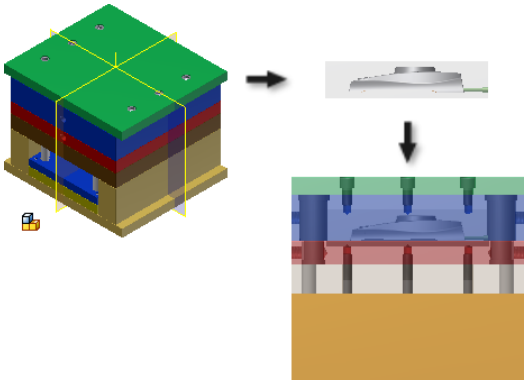
Patching and Runoff Surface Color



You can control the color of patching and runoff surfaces.

Before you create the core/cavity, right-click a browser entry, select Change Display Color, and set the individual or group surface color. The assigned colors persist when you create the core/cavity.

User Mold Base

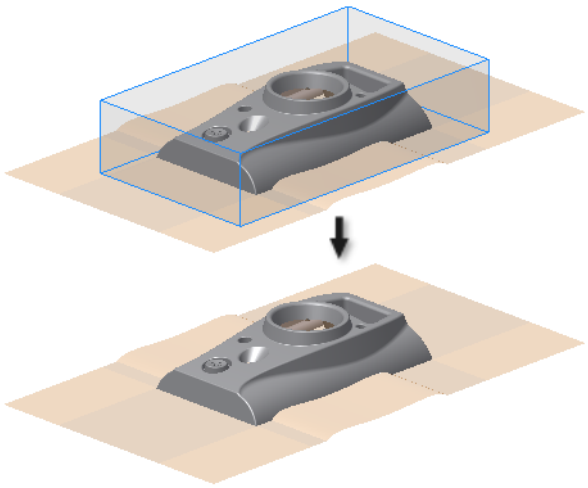


Create an assembly, and use the User Mold Base command to place as a mold base. Before placement:

- Use options in the dialog box to enable or disable automatic constraints.
- Use the Mold Base Author command to specify the component type and alignment.

If necessary, use the author command to define the mold base after placement.

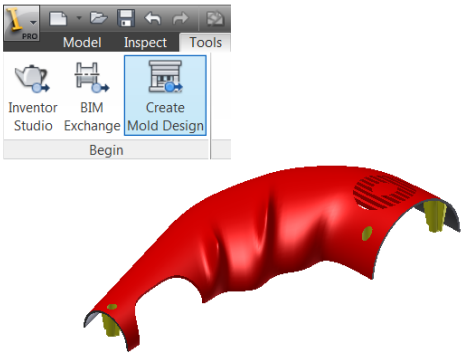
Edit Moldable Part



The command Edit Moldable Part, on the Core/Cavity tab, Parting Design panel, activates the moldable part file (MP) and turns off the visibility of other components.

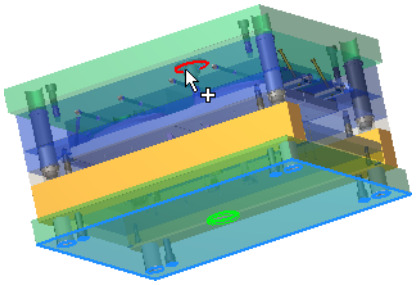
Choose this command to edit the body of the moldable part, and create hole patching and runoff surfaces in the MP file.

Transition from Part to Mold



Create Mold Design on the part Tools tab, Begin panel provides a smooth transition between part modeling and mold design. Choose this command to enter the mold design environment directly from the source part file.

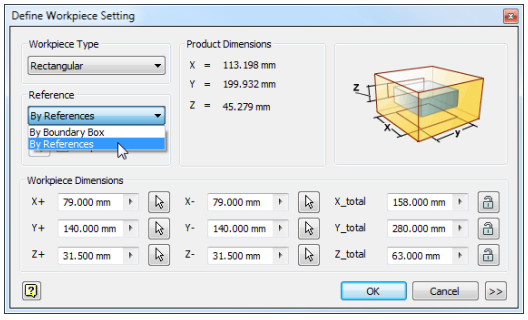
Multiple Locating Rings



Select a concentric reference and place multiple locating rings in the same command.

To place multiple locating rings, select the base face, a concentric reference, and then choose apply.

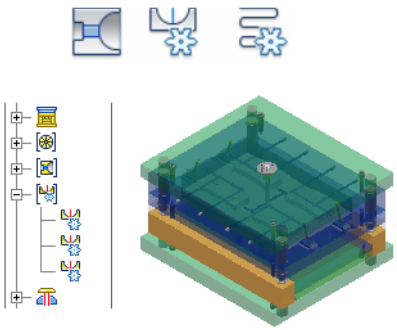
Define Workpiece Setting Enhancement



An option, By References, is added to the Define Workpiece Setting dialog box, Reference drop-down menu. Use the new option to:

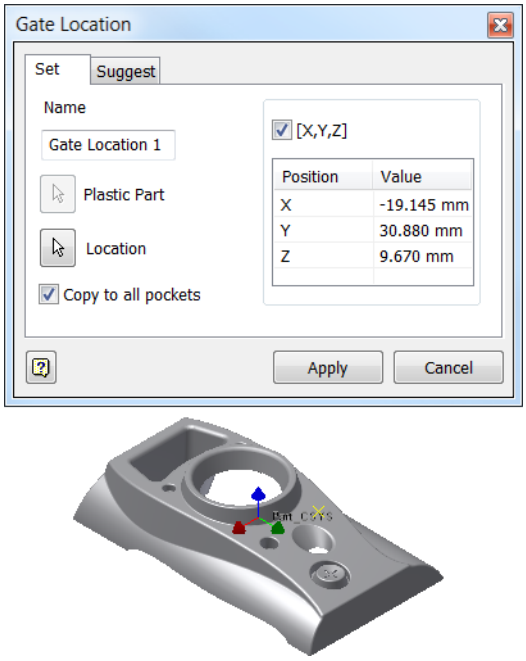
- Define a workpiece based on selected elements of the plastic part.
- Define a workpiece based on multiple selected elements of the plastic part. Turn on Multiple references to select more than one reference.

Retain Last Input Value



The commands Gate, Cold Well, and Cooling Channel are enhanced. The dialog box for these commands retains the last input value for the current session.

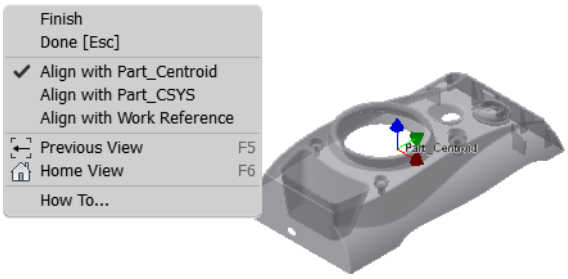
Gate Location X,Y,Z



An option to display and edit the gate location as X,Y,Z values is available in the Gate Location dialog box.

- To display or edit the value as X,Y,Z coordinates, select the X,Y,Z check box.
- To display and edit the position as U,V (ratio on an edge) values, clear the check box.

Place Plastic Part



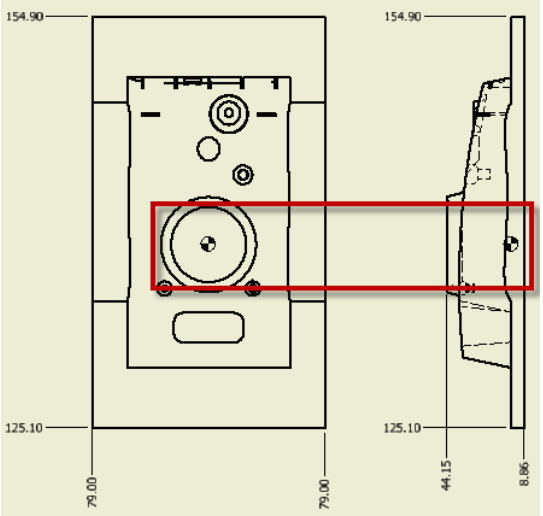
There are three alignment choices for the Plastic Part command.

- Align with Part Centroid (default)
- Align with Part CSYS (coordinate system)
- Align with Work Reference

NOTE You can select a UCS or a Work Point to define a Work Reference.

Before you place a plastic part, right-click to access the alignment options. Left-click to accept the selection and place the part.

Ordinate dimensions



You can now use the mold or part coordinate system as the origin for ordinate dimensions. Right-click when you place the plastic part in the mold assembly to specify which coordinate system is used for the mold. Enable Origin of Coordinates in the Drawing Settings dialog box to use the specified coordinate system.

Index