| Replace Face inputs | Results |

Benefit: This is a powerful and flexible functionality that allows you to capture design intent from organic surfaces. Simple prismatic solids can quickly be modified to become aesthetic shapes, significantly cutting down on design time. The ease of use offered from not having to extend the surface or worry about adding, subtracting material illustrates the typical ease of use Autodesk Inventor offers in its solutions.

Delete Face/Lump

Delete Face can be used to remove unwanted faces. It is possible to remove faces from both solid models and surface models. When deleting faces from a solid model, two special options are available, Healing and Lump selection. Healing closes the model, patching the solid model. Lump allows free-floating or disjoint solid regions to be deleted without having to select all the faces manually. Delete Face can be used with or without Healing. When used without Healing the solid model is converted into a surface model. This is easy to identify as the top icon in the browser switches from a solid cube to a swept surface. Once a solid is a surface, the surfacing tools in Autodesk Inventor can be used to edit its shape. Finally, using the Knit tools new in Autodesk Inventor 6, the solid can be converted back to a solid. This type of hybrid modeling relies on unified shape technology, a benefit of the Autodesk ShapeManager solid modeling kernel.
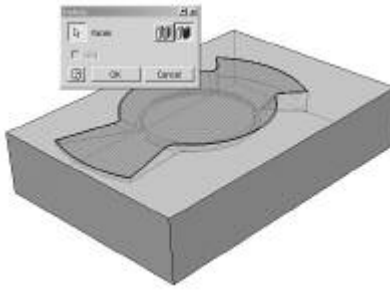
Delete Lump
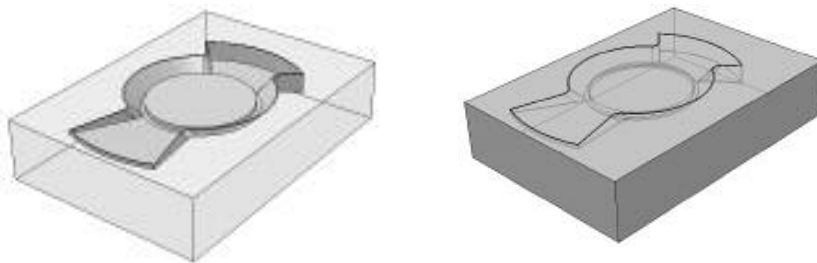
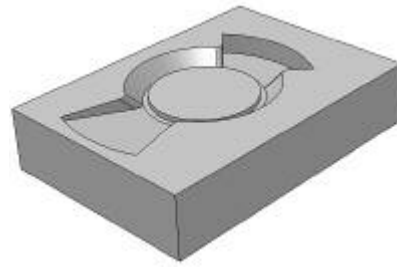| Assembly with plastic part intersecting block | Final part showing plastic part subtracted from block |

Delete face selecting floating
solid lump

Delete face selecting floating
solid lump

Emboss

The Emboss command wraps any sketch profile around a selected set of solid model faces. Emboss is designed to make it easy to create the text found often on plastic and thin metal parts. Like most Autodesk Inventor modeling commands, it is possible to do this in an operation that adds or subtracts material. Special provision is also provided for cases when part of the emboss adds material and part removes material. When creating an emboss, draft and direction are user defined. Because embossing is often used in aesthetic features where the embossed surfaces are painted or machined, the Emboss command allows faces to be automatically colored to an alternate color style.
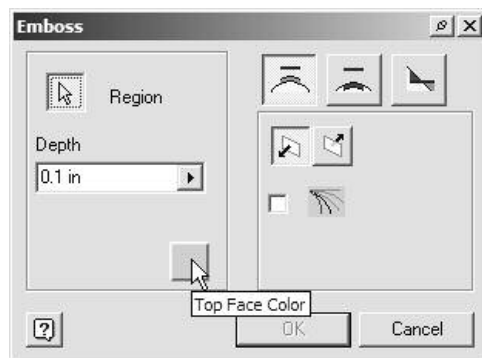
Emboss dialog box

Text embossed on a face

Benefit: Commonly used in plastic parts, the new embossing tools are an all-purpose command to cover all embossing needs. This offers you the tools and productivity needed to make creating plastic parts fast and easy.
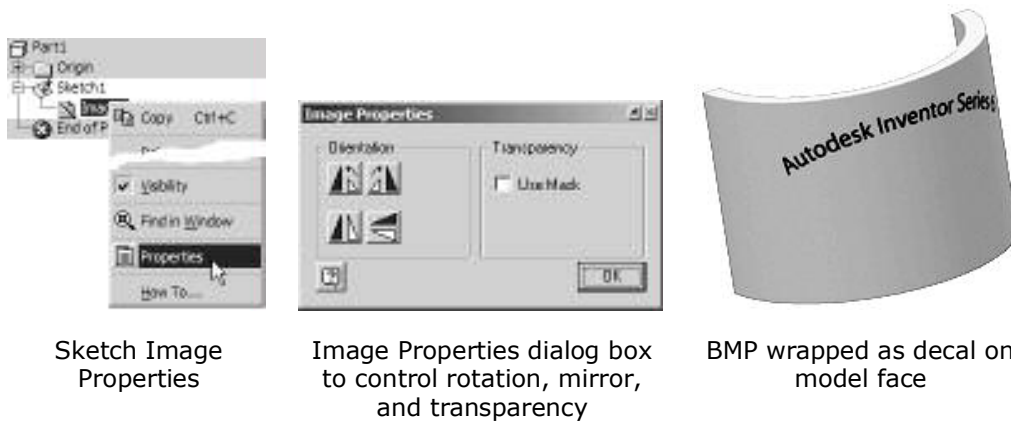
Decal

The Decal command can be used to convert images in sketches to an image that wraps to surfaces on a design. To use, place an image into a sketch. It should be dimensioned or constrained into position. Using the Image Properties dialog box transparency, rotation, and mirroring can be controlled. The Decal command can then map the image onto faces of the model, projecting the image perpendicular to the sketch plane or wrapping it perpendicular to the surface.

To make portions of the image transparent, activate the mask color checkbox. The transparency color is determined by the lower right pixel of the image—all pixels of this color in the image become transparent, similar to material textures.

Sketch Image Properties	Image Properties dialog box to control rotation, mirror, and transparency	BMP wrapped as decal on model face

Benefit: Often designs are marked with bar codes, labels, or decals to help identify them. Decals in Autodesk Inventor 6 provide a fast easy way to accomplish this.

Pattern along path

When defining a rectangular pattern, sketch geometry or model edges can now be used to define the path directions. Using paths allows you to control the start point, spacing, number, fit, and direction of pattern occurrences along the path.

Paths can be any model edge, 3D sketch or chain of sketch entities. Paths do not need to be tangent-continuous. A pattern can be created along two paths in different directions using any combination of inputs.

Each path has a start point to derive occurrence position relative to the path. Each path also has a direction, which can be switched to meet design needs.

When patterning along a path the option to fit to path length automatically calculates the spacing between a given number of occurrences.

Dialog	Options on the More button
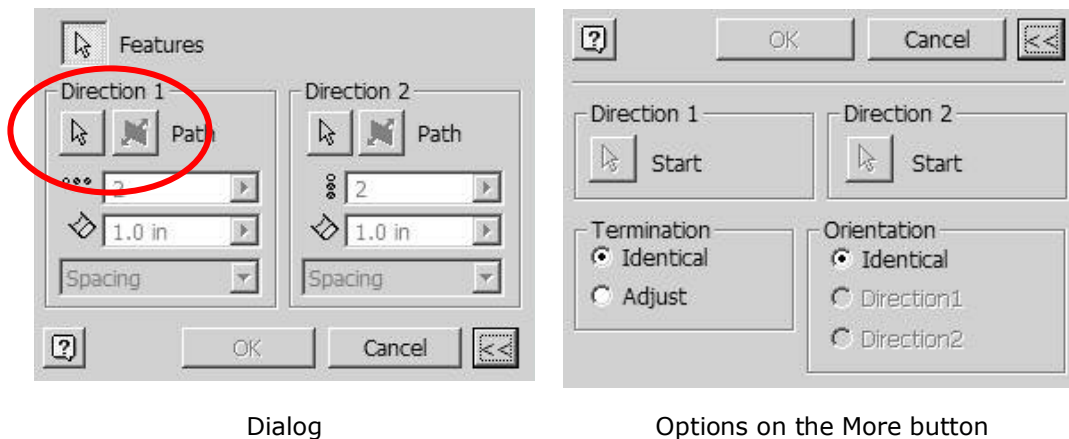
Benefit: The ability to use a path to define a pattern opens entirely new pattern shapes not previously available.

Surface construction environment

Autodesk Inventor uses a construction environment to perform a Hold operation on surfaces and prepare them for use. The construction environment allows you to stitch individual

surface together, evaluate the quality of edges between surfaces, and then promote these surfaces into the main modeling environment for use.

Autodesk Inventor 6 now allows the contents of the construction environment to be saved and used later even if the entire part file is closed. It allows large amounts of imported data to be used incrementally over time without needing to repeat the import each time the file is opened.

Stitching (also called knitting) surfaces in the construction environment now provides tools to analyze the edge quality in the resulting quilts. This is important for spotting gapes or tolerances errors in these surfaces. Visually communicating the problem areas helps you focus on problem areas so that you can make high-quality surface data for import.

Once you're satisfied with surfaces or newly created quilts in the construction environment, Autodesk Inventor then allows the data to be promoted into a Grounded surface in the main parametric modeling environment.
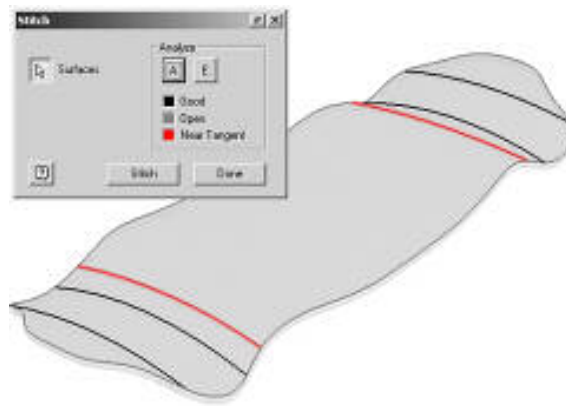
Stitch showing edge quality on 3D surface model

Benefit: Exchanging surface data is common in many industries. By offering better tools to use and integrate these surfaces into designs Autodesk Inventor gives you a way to reliably use and change surface data, with the advantage of having higher quality results and lower cost of changes.

Model Tolerances

Nothing can be made to an exact size. When a part is manufactured, there is a range of acceptable values, or tolerances, for each dimension. A basic problem in design is determining the size of that range. The more precisely an item or feature must be made, the more expensive it becomes. The functionality of the part can also be impaired if the tolerance range is too great.

Tolerance analysis occurs at several levels. For example, assemblies need to be checked to ensure that they can be assembled and that there are no interference issues when parts reach the extremes of their tolerance ranges.

Model parameters in Autodesk Inventor 6 can have a tolerance value that can be retrieved in the drawing. You have three methods for setting the tolerance:
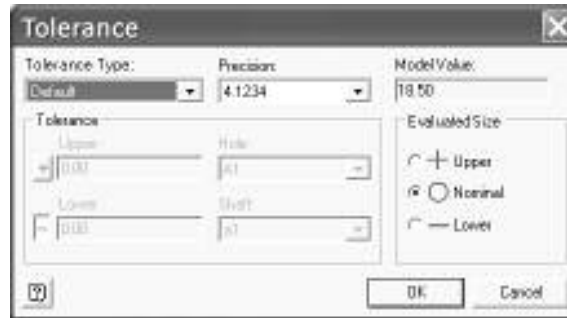
- Edit the dimension

- Edit the feature

- Setting in the parameters dialog box

When editing a parameter from the Edit Dimension dialog box, or a input field in a Feature dialog box, the context menu of the edit field holds an new entry called Tolerance. If this is selected the Tolerance dialog box appears.

Tolerance dialog box

When using the Parameters dialog box in a part, you have visual cues to the condition of each parameter that display if the parameter is at Nominal, Max, or Min values. For each parameter you can use the Tolerance dialog box to edit the type and value of the tolerance.

For fast and easy analysis you can reset all tolerances to a given condition from the Parameters dialog box, using a set of buttons to set the global condition in one step.
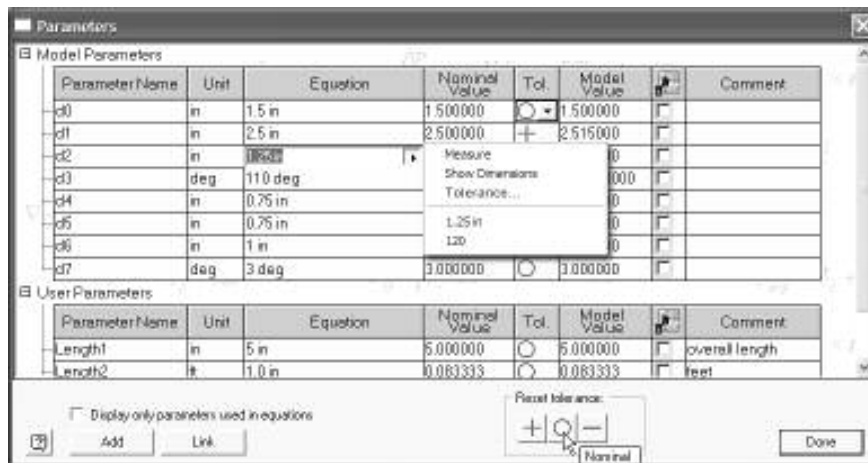
The updated Parameters dialog box

Benefit: By capturing tolerances at the time of design, less information is lost between design and drawing documentation. Using tolerances to calculate material conditions also increases your ability to explore the effects of tolerances on your design. This reduces mistakes and helps you to know and capture more about the design.

Face Colors

In addition to whole part and feature colors, you can now select individual faces and apply unique color styles. A face's color can be set to any specific color style in the part, or it can inherit its color from the feature color or the part color.

**Benefit:** When applying color styles, you can easily do the job using face colors. When textures represent processes like knurling or machining and do not coincide with single features, they require face-by-face color control tools. The result is that your models look more appealing and are suitable to be used in marketing or promotional materials.

Mass Properties Overrides

When working with mass properties, mass and volume can now be overridden. Values are displayed with icons to clearly show which are calculated from the solid model and which are overridden.
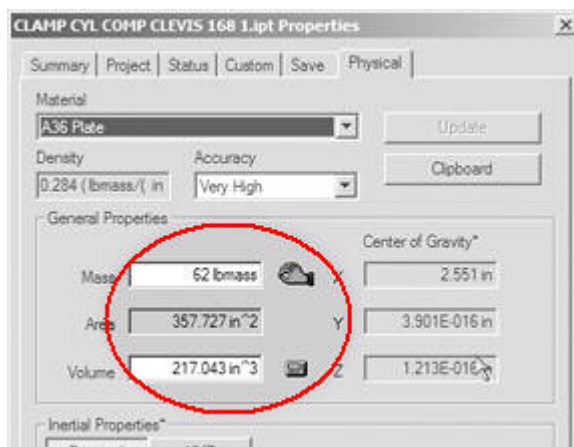
Icon for calculated value

Icon for overridden value

Benefit: Often a simple envelope model is all that is required to represent a purchased component. When using these simple envelopes, the mass properties of the solids do not accurately represent the weight or volume of the true part. Now you can manually enter these values ensuring accurate mass properties. This ensures the ability to precisely analyze designs for weight and center of gravity.

Sheet metal

Measure in flat pattern

The Measure tool can now be used in the flat pattern window for sheet metal parts. This allows unfolded distances to be easily checked.

Benefit: Measure tools are quick and simple way to verify or find important information about the size and shape of your designs.

Suppress bend zone lines in flat patterns

When making a drawing view of a sheet metal part, there is a new checkbox option on the view properties. This option is similar to Show Tangent Edges, but it controls the display of the bend extent lines in the view.

Bend extent lines and bend lines are now in the list of line types in the standards manager for drawings. This gives you control over line style, weight, and color.
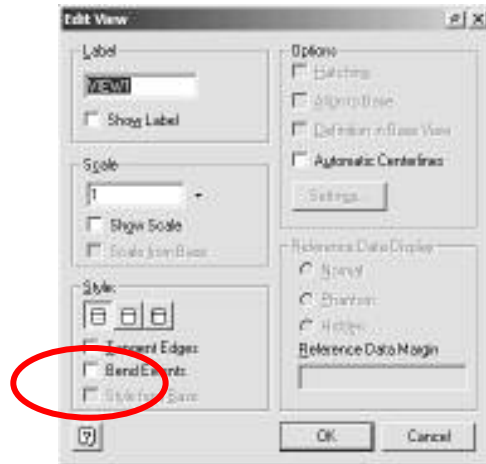
Benefit: Removing bend extent lines gives you more flexibility to format drawing views. In addition when exporting the view to DWG or DXF™ for fabrication, this geometry (which causes some CAM systems difficulties) can be removed.

Repair holes in flat pattern

When holes are cut across bends, the flat pattern analyzer interprets them as splines. Improvements were made in replacing these splines with circular holes.

Add none as a bend relief shape option

Some sheet metal shops avoid the use of bend reliefs since it adds cost. This option allows designers to set none as the default and add bend reliefs only when required. This improves the workflow and productivity.

Sketching

New Sketch Constraints Captured

When you are sketching in Autodesk Inventor, the constraint system now infers an additional constraint—concentricity—on the fly.
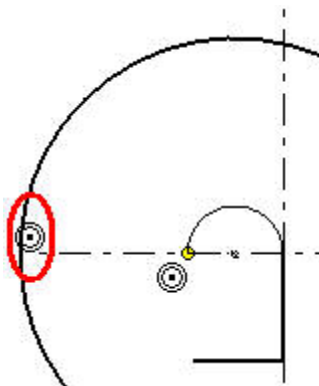
Concentricity

Benefit: The sketching environment in Autodesk Inventor software exemplifies intelligent, gesture-based design. By accepting inferred constraints on the fly, you spend less time adding constraints manually.

Midpoint Sketch Object

Sketch lines and arcs now have a selectable midpoint. This allows constraints to be applied to the midpoint of these objects without having to draw a midpoint point first.

Benefit: It's often good mechanical design to create symmetric parts to allow left- and right-hand use without needing different parts. Autodesk Inventor now allows easier constraining to the midpoint of a sketch line or arc without having to create the point first. This makes constructing symmetrical parts faster and easier.

Project Loop or Chain

Entire loops can now be quickly projected while in Part Sketch mode. Projected loops are fully associative to the parent face and topological changes are easily maintained. For example, a rectangular projection of a slot inherits the topological changes from the filleting operation on the parent slot. A face selection option is also available that projects all loops contained within the selected face, both outer and inner loops.
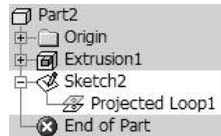

Browser showing sketch with
projected loop

Benefit: The ability to quickly select one loop element instead of numerous elements dramatically reduces design time. This also increases the robustness of sketch-based features as the entire face is associative instead of just a few segments that were projected.

Offset Splines and Ellipse

The Offset sketch tool can now offset a spline or ellipse. The offset does not allow the offset shape to collapse.
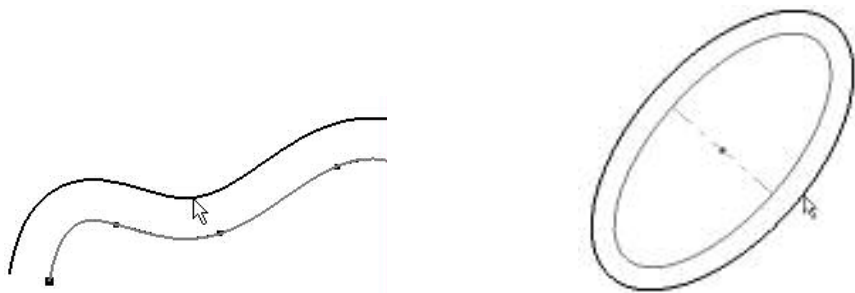
Offset spline   Offset ellipse

There are two methods to offset an ellipse. Select the ellipse near the major or minor axis to construct a concentric ellipse. Select the ellipse in a location away from one of the axes to construct a spline profile. This allows you to easily construct the required type of offset ellipse.

Benefit: Consumer products often use splines to define complex shapes. Being able to offset splines makes the job of defining consumer product type shapes much easier.
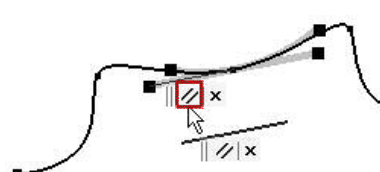
New Spline Controls

Splines are updated for easier editing and control. Specific improvements allow you to interact with the knot points of a spline and define radius of curvature and curve influence. You can now control overall spline tension and use spline knot point handles as objects to which constraints and dimensions can be applied.
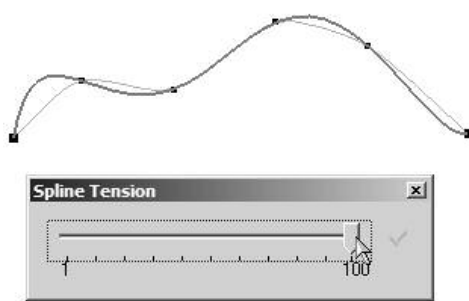
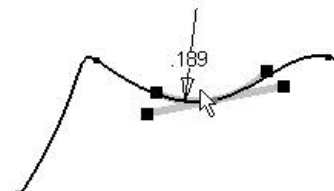Spline tension control slider to control shape
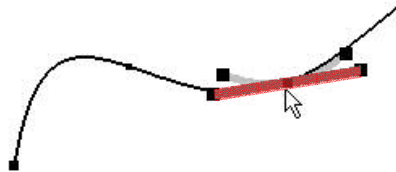
Bowtie manipulator to edit spline control point influence

Bowtie can be used for sketch constraints/dimensions

Curvature bar to edit spline control point radius of curvature

Curvature bar can be used to apply constraints/dimensions

Benefit: Splines are frequently used to define the wireframe geometry from which complex consumer product shapes are derived. More complete control over shape and position of splines ensures that you can create the exact shapes needed.

Sketch Text Profiles

The part modeling sketch toolkit now offers a sketch text command. This creates text which can be used as profiles for other modeling commands. Text might be used in emboss, extrude or other sketch profiles based features.

3D Sketch Constraint

This initial 3D Sketch constraint creates a coincident constraint between two points within a 3D Sketch. Possible combinations include:

- Sketch point / Vertex
- Sketch point / Sketch point
- Sketch point / 3D Intersection Curve

Benefit: 3D sketch constraints can be used to create associative relationships between various 3D sketch objects (similar to constraints in the 2D sketch environment).

3D Sketch from Intersections

This 3D curve creation technique is based on the intersection of freeform surfaces, work planes, or faces. This 3D sketch can then be used to create a complex sweep feature or used in other features such as loft or aligned pattern.

Benefit: 3D curves can be used to define complex shapes often seen in consumer products. The 3D curve is one of the underlying pieces of geometry used to construct these organic shapes.

Assembly Modeling

Assembly Features

Assembly features are features that are defined in an assembly. They only affect a part when the part is viewed in the context of the assembly. Assembly features allow you to describe specific manufacturing process with Autodesk Inventor, such as match drilling and post-weld machining. The assembly features you need most often can now be easily described with the new assembly modeling features added in Autodesk Inventor 6.

Assembly Panel showing assembly
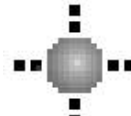
features

Assembly features
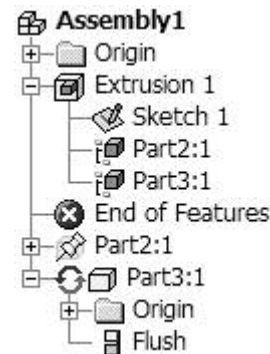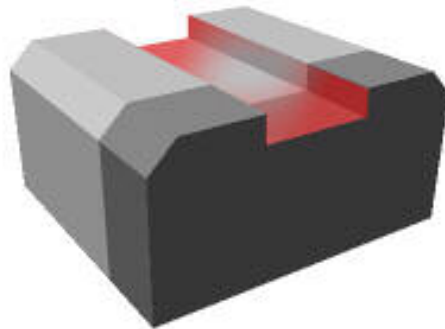
| Hole | Cut | Chamfer | Work plane | Work axis | Work point |

Post-machining operation example

Match drilling operation
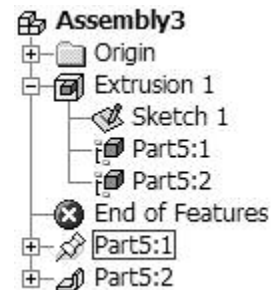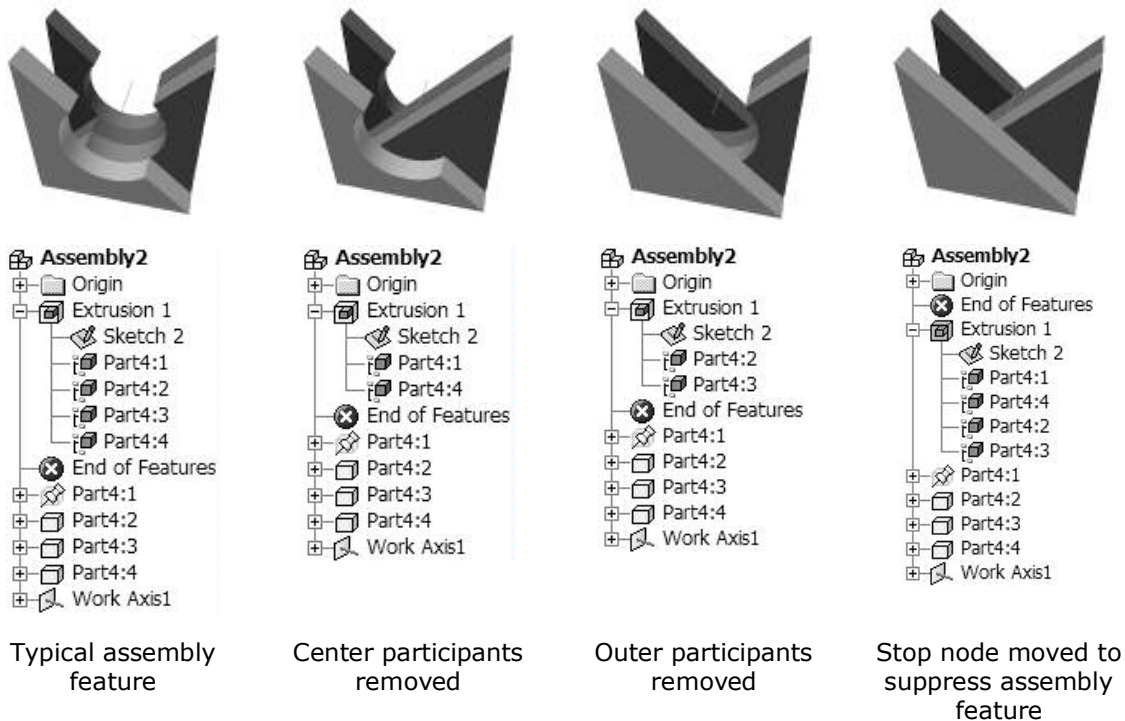
When working with assembly features it is possible to add and remove parts that are affected by a feature. These affected parts are called participants and appear as children of the feature in the assembly browser. To remove a participant, right-click the feature and select Remove participant. To add a participant, select Add participant, then select the part in the graphics or the browser.

| Typical assembly feature | Center participants removed | Outer participants removed | Stop node moved to suppress assembly feature |

Benefit: Assembly features allow you not only to model geometry, but to build and capture manufacturing process and engineering. Now it's easy to design and document post-machining operations, match drilling, and other processes common in industrial machine and electro-mechanical designs.

Design for Welding

A new task-based environment in Autodesk Inventor 6 accelerates design of weldment assemblies. The weldment environment encompasses the part design of the weldment assembly as well as weld bead specification, weld edge preparation, and post-weld machining. The Autodesk Inventor drawing manager enables you to create drawings that document the entire weldment manufacturing process, from component drawings to pre-weld drawing views and post-machining views. Special 2D symbols allow for both automatic and manual creation of standards-based welding drawings. This complete workflow of 3D design to 2D documentation ensures that all the design information captured in 3D transfers to the 2D drawing automatically.
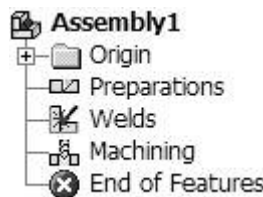

Weldment browser folders

As weldments are built, the welding environment collects features in folders to organize the design. Special folders in the assembly browser hold prep features, weld beads, and post-machining operations. Each folder can be activated by a simple double-click, allowing you to

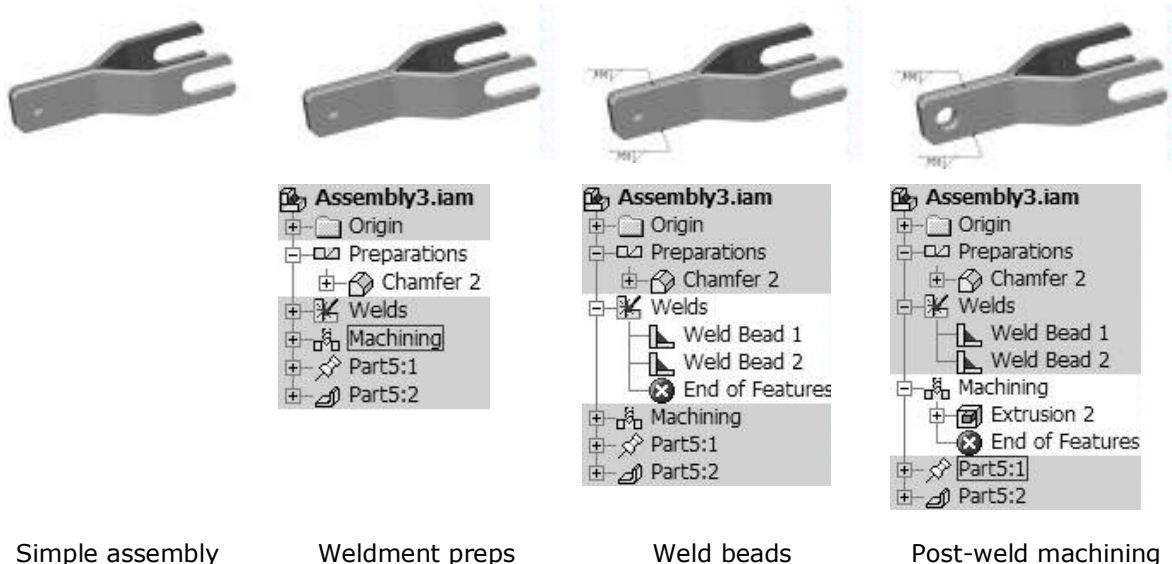add to and modify its contents. In this way the complex process of designing a weldment can be managed easily.
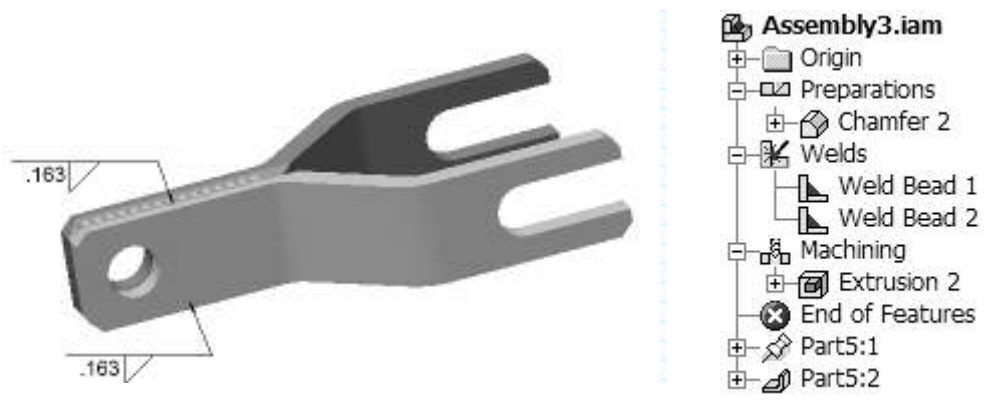
A simple sample weldment

| Simple assembly | Weldment preps | Weld beads | Post-weld machining |

Weld preps can be made using extrude cuts and chamfers. They can be captured at the assembly level using the new assembly features in Autodesk Inventor.

Weld beads offer you flexibility in choosing both cosmetic and physical methods of defining welds; that is, creating welds as a graphical element or as a realistic 3D body. This choice allows the weldment design process to fit different design needs. Cosmetic welds offer excellent performance, the widest range of weld types, and fast easy definition. Where interference detection or mass properties effects of welds are a concern, physical welds are the better choice.

Cosmetic welds are annotations and symbols that identify the edges to be welded. The chosen edges are highlighted and a 3D symbol, pointing to the edge, calls out the specifics of the weld type.

When creating a drawing of a weldment, the dialog box offers special options. The view can be defined to show the weldment in any one of four steps of the welding process: the

assembly alone, the assembly plus preps, the assembly plus preps and welds, and the assembly plus preps and wells and post-machining operations. In addition to the weld types, you can also elect to automatically retrieve weld symbols and annotations.

Constraint Tools

Assembly constraint management has new tools to help you track information about constraints. First is a new "Other Half" tool in the context menu that allows quick access to the matching part participating in the constraint. Second is a new tooltip that displays all property information—including constraint type, offset value, direction, and part names affected—for the highlighted constraint. Finally, constraint offset values are now modified through the standard value edit control. This more consistent method allows you to measure while editing constraints through the familiar dialog used to edit other values like work plane offsets or sketch dimensions.
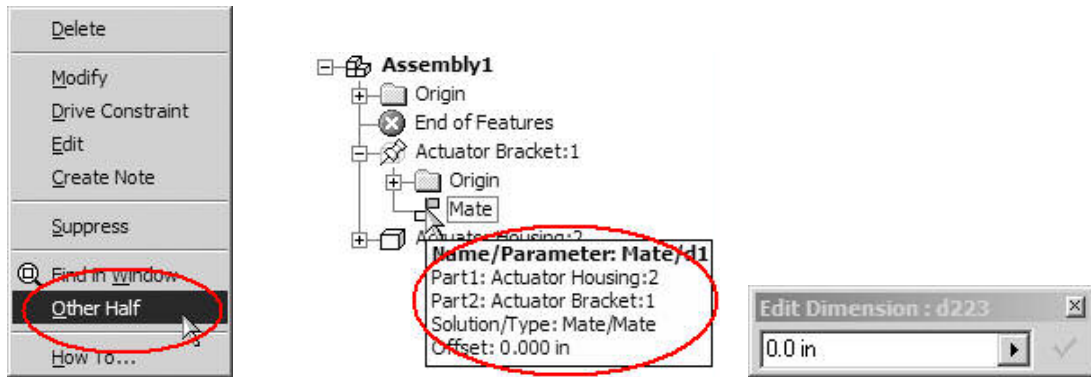
| Other Half tool locates mating part quickly	Assembly constraint tooltip	Constraint value dialog box

Benefit: Autodesk Inventor 6 provides easy-to-use tools that give you high-feedback information about constraints. These enhancements directly address your needs as you work with and maintain constrains on assemblies with thousands of parts. These enhancements ensure that Autodesk Inventor has the easiest to use tools for assembly design.

iMate Publish

IMates provide an easy way to store and reuse positioning data. To make creation of iMates easier, you can now extract existing constraint data to convert it to iMates. This process allows iMates and Composite iMates to be generated from the constraints on the one part or from all constraints of all occurrences of the selected part in the active assembly.
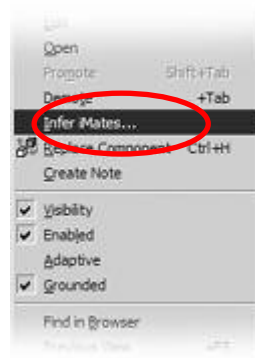
Infer iMates tool converts constraints	Infer iMates options

Benefit: Creating and cataloging placement information is now fast and easy. Companies can begin to capture the knowledge of how components are assembled and reuse it to save time and reduce errors.

Adaptive Parts

Adaptivity now allows for the inclusion of projected and fixed geometry. Many users migrating from previous CAD systems developed the habit of always constraining the sketch of the first feature in their models to the origin. Autodesk Inventor now allows such parts designed with this technique to be used adaptively in an assembly.

Benefit: Fixing the first feature to the origin is a good modeling technique that makes model changes easier and more predictable. Allowing designs to be adaptive gives you a powerful tool to solve for form and fit with the benefits of reliable and predictable part updates.

Assembly Update Button

The Update button now reflects when an assembly needs updating. When the button is inactive, you know that the current assembly is fully up-to-date. When it becomes active, you know that there are pending updates that need to be processed.

Update ▾	Assembly needs updating
Update ▾	Assembly is up-to-date
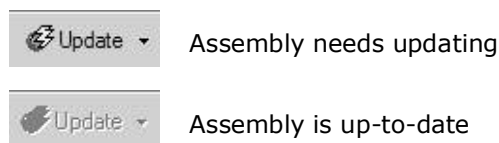
Benefit: The new button behavior helps you manage updates, which can be time-consuming operations. In addition, the Update button now behaves the same here as it does in part modeling, which helps make Autodesk Inventor easier to use and learn.
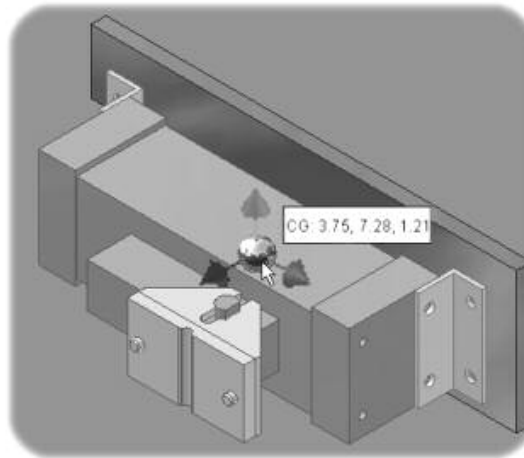
Center of Gravity

Selecting the View menu and choosing Center of Gravity toggles the display of the assembly's center of gravity.

Assembly View menu   CG Indicator tooltip

Moving the cursor over the glyph causes the center-of-gravity coordinates to display in tooltip fashion. When a component is out-of-date, the program asks if the component should be updated. If you reject the update, the glyph displays in "ghost" mode. If a part or an assembly is out-of-date, the glyph displays in the last known location.

Updating mass properties refreshes the center-of-gravity location and glyph display. To make updating mass properties easier, the function is now located on the Tools menu for parts or assemblies.

Design Views and Visibility Overrides

Autodesk Inventor enables you to globally override the display of origin work planes, origin work axes, origin work points, user work planes, user work axis, user work points, and sketches within an assembly. Turning off unneeded geometry reduces screen clutter and improves graphics performance. Now these overrides are stored as part of the design view.

Benefit: Because design views are specific to each assembly file, display overrides are now remembered on a document-by-document basis, removing the need for each user to have to manually set the desired display overrides for a given assembly or design.

Promote from assembly

When editing a part in the context of an assembly, the Promote tool is now available to copy surface or solid geometry between parts. The copy is nonassociative, but it allows for the copied geometry to be used for reference.

As an example, an extrusion needs to be terminated to a surface of another part. This termination is a lofted surface and is not planar. Using Promote, you can select the part in the assembly, then the needed termination surface, and Promote it by copying it into the active part as a grounded surface. This copied grounded surface can be used as the termination plane. The position of the copied geometry is defined by the current position in the assembly from which the Promote operation was performed. Should the surface need to be changed, you can use the Redefine option to select a new or changed surface.
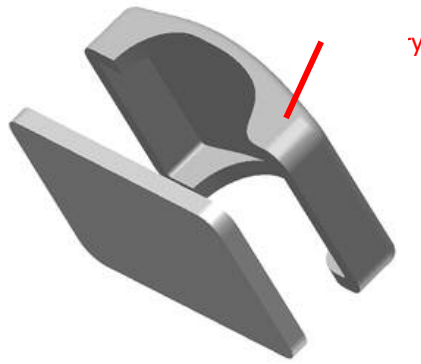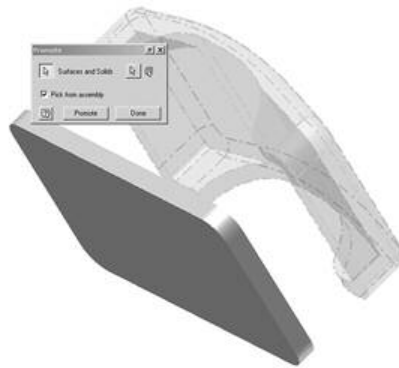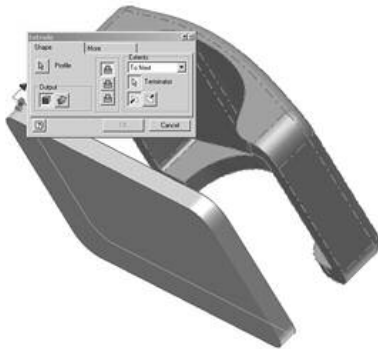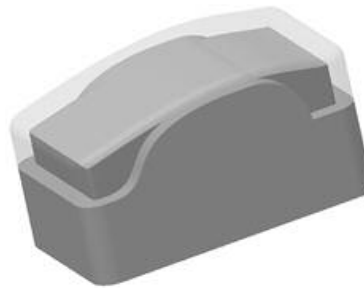
Geometry needed as termination
surface

Promote selects the part, then the
geometry

Extrude can now terminate to the
surface

Finished extrusion

Benefit: Using adaptivity, parts can be resized to fit design requirements. Using cross-part projection, sketch geometry can be created using edges from other parts. And now using Promote, surface or solids geometry can be copied between parts. These tools reduce the work required to create "top-down" designs.

Component Pattern Replace

When performing Replace operations on an assembly pattern, assembly pattern occurrences are now replaced with the newly selected component. This includes selecting a new iPart configuration.

Benefit: Using component patterns allows designs to better capture design intent. Replacement of all occurrences maintains this design intent without making you replace each component in the pattern manually. Overall assembly ease-of-use is improved, and working with component patterns can be completed faster.

Component Pattern Restructure

It is now possible to restructure a pattern into or out of a subassembly by using the restructure tools found on the context menu in the assembly design environment.

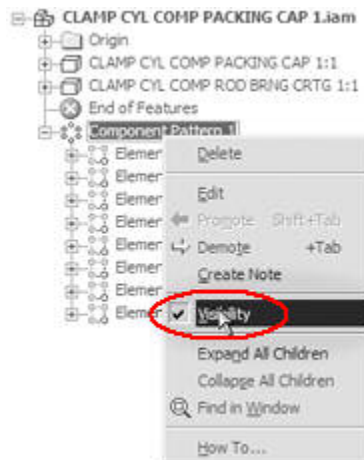**Benefit:** Restructure allows you to focus on design first, then move parts into a logical structure later. Allowing patterns to be restructured allows you more freedom when arranging this logical assembly structure.

Component Pattern Visibility

Visibility of a component pattern in an assembly can now be toggled from the context menu of the root pattern node.

Benefit: You no longer need to expand the pattern and select each pattern instance to toggle the visibility of patterned components off. Now it's easier to turn off parts not needed for immediate design tasks, helping reduce screen clutter while improving graphics performance.

Cancel Open on Resolve Failure

When working with many projects, you may sometimes forget to activate the correct project to open the assembly. In Autodesk Inventor 6 you can cancel out of the Open operation should a resolve failure occur.
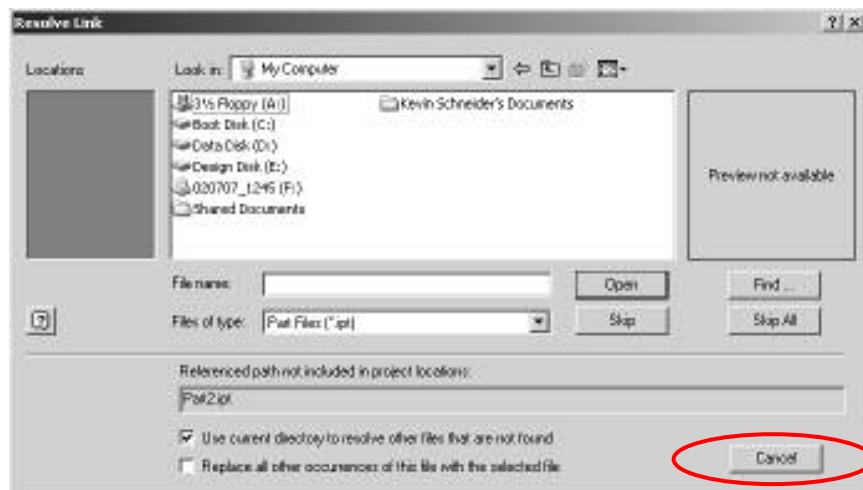
Cancel allows getting out of a bad situation

Content Library

www.autodesk.com/inventor

Libraries

Autodesk Inventor 6 adds a new comprehensive standard parts library with several enhancements and improvements over past parts libraries. The library now supports 18 international standards and up to 12 different classes of standard parts: screws and bolts, nuts, washers, adjusting rings, bearings, circlips and O-rings, parallel keys, pins, rivets, roller bearings, shaft seals, and woodruff keys. The types of parts have been expanded to add more shaft parts like retaining rings, bearings, and woodruff keys. In all there are several hundred thousand individual parts possible.

When working with the Library you can now search, view a placement history, and keep a list of favorite parts to make working with the libraries fast and easy.
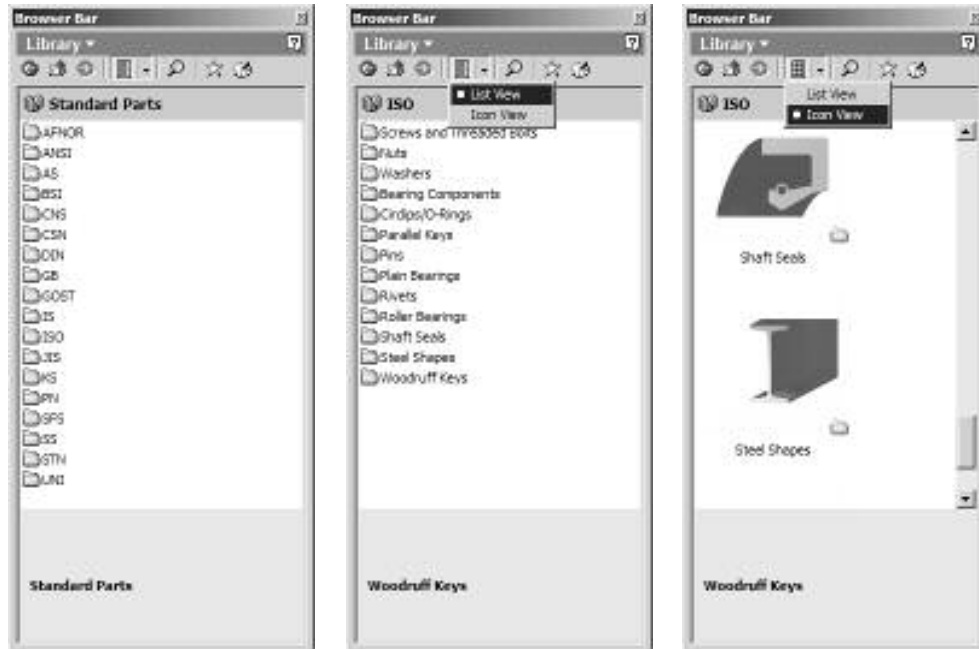
Available standards	List View	Icon View

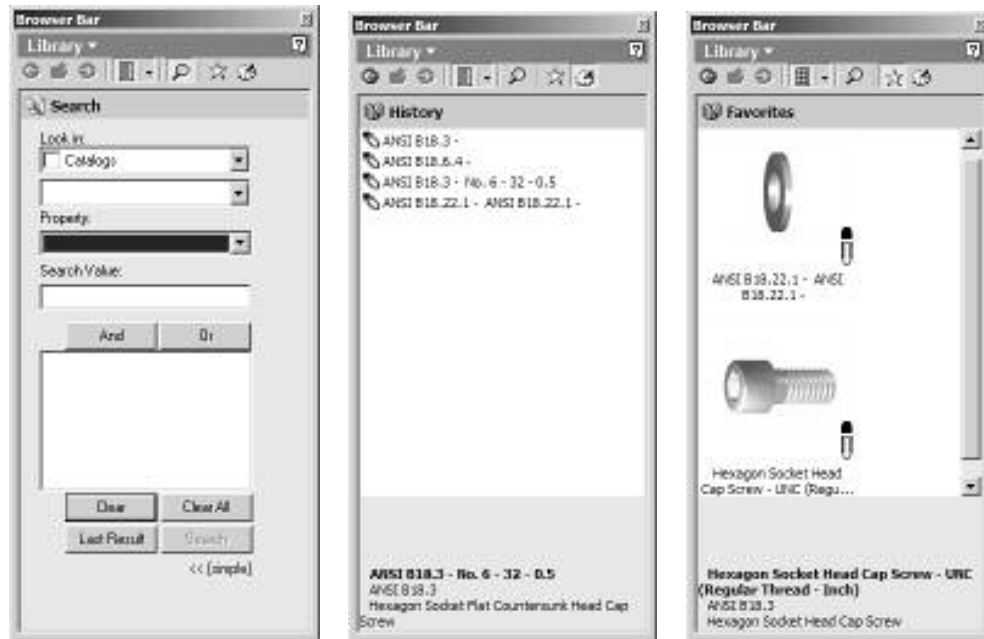|		
Search View	History View	Favorite View

Should a library part need to be changed or replaced, a new context menu locates the part in the catalog automatically and allows it to be replaced for a new size or type.
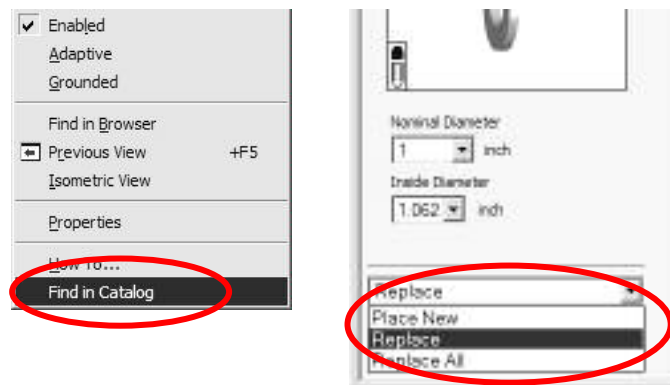
Assembly context menu	Replace tools

Benefit: As much as 80 percent of a typical industrial machine can be made of standard and purchased parts. Autodesk Inventor 6 offers a comprehensive library to make it easy to design and use standard parts quickly. Frequent users will find the favorites, searching, and history views make working with the library even faster and easier.

Interoperability

MDT translator update

Several enhancements in the Autodesk® Mechanical Desktop® translator expand the quantity and capability of features that are imported to Autodesk Inventor software.

- Cut, the most commonly used option of the Surfcut command in Autodesk Mechanical Desktop, is supported in Autodesk Inventor 6. Surfcut features are translated as fully editable Split features in Autodesk Inventor.

- With the improved color enhancements, Autodesk Inventor preserves BYLAYER coloring of Mechanical Desktop components when translated. This allows you to easily recognize components graphically after translation.

- Rib features from Mechanical Desktop are now translated as Rib features in Autodesk Inventor with the source sketch being fully editable.

- Additional termination types for sketched features are supported. These include Work Planes and additional Face conditions such as Extended Face.

DWG import

AutoCAD layers to sketches—Autodesk Inventor now translates each layer that contains data into individual Autodesk Inventor sketches. This feature imports the data into either part or drawing files. By doing this the data is more organized and helps preserve the original intent of the data. Bringing the data into many sketches also helps speed up the importation of large AutoCAD® files.
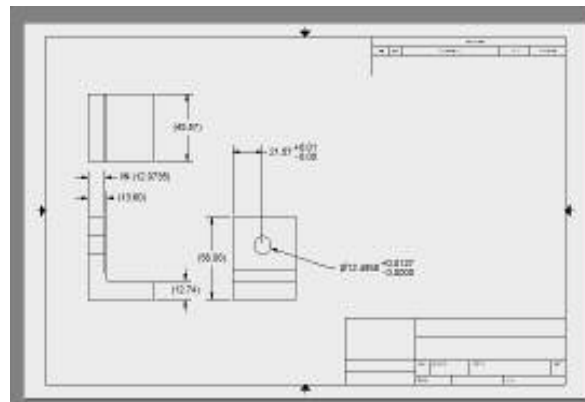
Import dimensions into sketches—AutoCAD dimensions are now placed on the same sketch as the associated geometry. When an AutoCAD dimension object is encountered the geometry it is associated to is found, and the dimension is generated on the same sketch even if the dimension was on a different layer to begin with. This improves the workflow when incorporating AutoCAD data to create Autodesk Inventor parts. This will also avoid the use of the Promote operation to sketch as well as reduce the cut-and-pasting sometimes needed between drawing and part files.

Wizard update—Many new wizard settings are available for the importation of DWG files, so you can fine-tune the importation of data:

- Endpoint constraints for Mechanical Desktop drawing importation.

- Selecting optimized placement of layouts puts the left and bottom most data point as the origin point in a drawing. Your data is always in the right place for plotting.
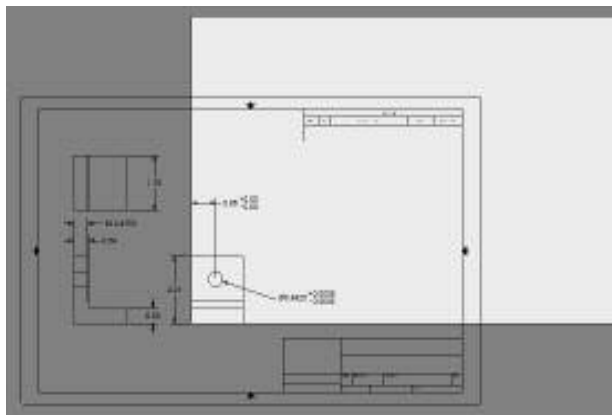
No optimized placement (Autodesk Inventor 5.3)   Optimized placement (Autodesk Inventor 6)

- Multiple selection of layers in the import wizard allows you to more quickly include or remove layers for translation. By default all the layers are selected.
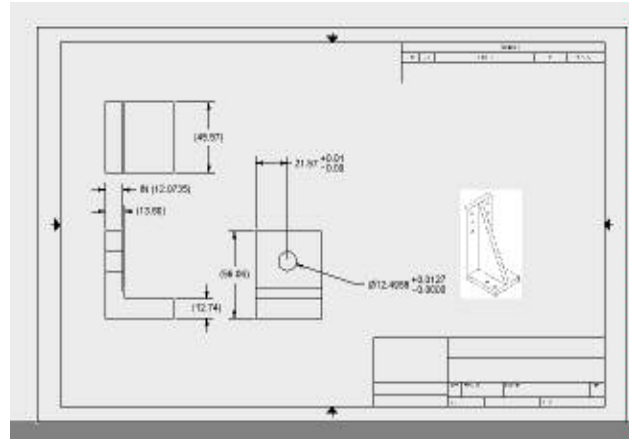
There are also some new tools for advanced users:

- Fonts from the AutoCAD file can be mapped to any font in Autodesk Inventor, for complete control over the fonts and how they appear in Autodesk Inventor.

- Control over double-stroke and single-stroke fonts is also available during translations.

- Not only are layers mapped to sketches, but you have full control over the names of the sketches being generated.

OLE objects Bitmaps
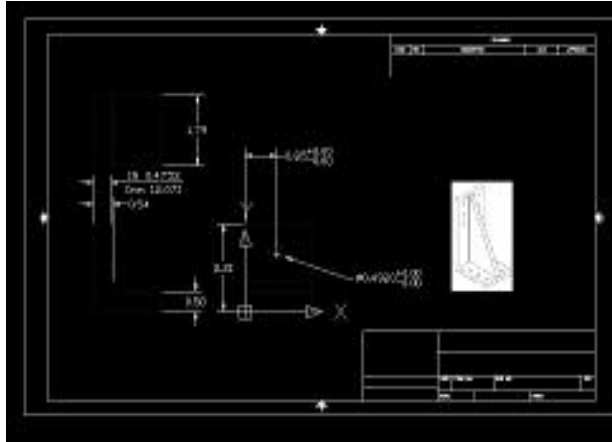
| AutoCAD file with bitmap | Translated bitmap object into Autodesk Inventor |

Multiple Layouts—Both model space and paper space can be imported simultaneously. You can translate a multilayout DWG file into a multisheet IDW file in a single operation.

Improved Leader import—AutoCAD leaders are imported as Autodesk Inventor leaders and are associative to the draft view should the view move or the scale change. Arrowhead types are also more closely mapped to the Arrowhead types in Autodesk Inventor.

DWG Export

Export symbols as blocks—Any symbols used on the face of an Autodesk Inventor drawing are exported as a block to AutoCAD. This allows you to insert symbols in AutoCAD for any editing that may be required. Title blocks with properties are also exported as blocks. Once in AutoCAD these blocks have Attribute tags and behave as they do in Autodesk Inventor.

Line type mapping—For advanced users, fine-tuning of line types is now possible during the export of drawing files to AutoCAD. You can map any Autodesk Inventor line type to any AutoCAD line type, even user-created line types. Line scales are also preserved during translation to help ensure expected results.

Sketch hatches and color fills—Sketch hatches and color fills are now translated to AutoCAD when exporting drawing files.

Explode Dimensions—Dimensions have always been exported to AutoCAD files as dimension objects but now you can export dimensions as exploded dimensions. When using this selection, dimensions are exported to AutoCAD as lines and text rather than AutoCAD dimensions. The dimensions created in AutoCAD are similar to those created while the DIMASSOC setting is turned off.

IGES/STEP import

IGES data now import even faster than before. IGES subfigure definitions blocks are now handled as components. These enhancements make large IGES files easier to work with and easier to use, because each block is translated into an Autodesk Inventor part file.

Multi-lump parts found within a STEP file are represented as such within Autodesk Inventor; there is an exact match between the STEP file and the Autodesk Inventor file.

Information found within the IGES or STEP file is now placed in the File properties Custom section. This information lets you know more about origins of the IGES or STEP file including the original file name, the sending system, and version.
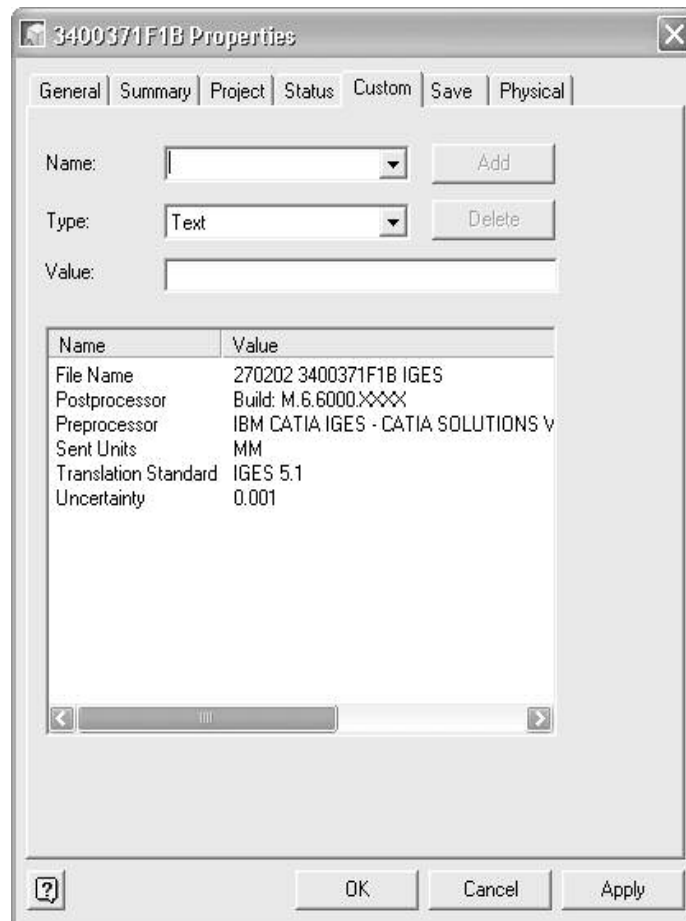
Autodesk Inventor translated file properties

Assembly features are fully supported in all the translators. IGES, STEP and Sat files do not support this feature within their specifications. However, Autodesk Inventor handles this information by changing the original part data as found within the assembly being translated. This allows Autodesk Inventor files to look the same in other CAD systems as they look in Autodesk Inventor.

Application programming interface

Autodesk Inventor provides a powerful and easy-to-use API that enables you and third-party developers to extend and customize the software. Autodesk Inventor 6 adds new capabilities to this API in the areas of drawings and UI customization.

Benefit: You can look forward to third-party add-on applications that integrate into Autodesk Inventor seamlessly, with the same look and feel and ease of use you expect from Autodesk Inventor software. You will save money by taking less training time.

Here are more details about the API enhancements.

Part Functionality

- Enhanced thread edit support
  - Tapped hole features
  - Thread features

- New feature query and support
  - 3D Sketch support
  - Sweep feature
  - Loft feature
  - Split feature

- Set position of stop sign—Control roll back to support a variety of application workflows, such as positioning the end-of-part marker to reconstruct the model and query the B-Rep at various stages of construction

- Access to work surfaces

- Enhanced iFeature support
  - Query and edit of associated parameters
  - Place an iFeature with access to all input parameters and geometry

- Enhanced iPart support
  - Create an iPart factory
  - Access to the table, setting current member, and placing members within an assembly

- Access to matrix that defines sketch to model transform

- Pattern feature enhancements

- Ability to unfold a sheet metal part

- Access to the B-Rep of the sheet metal flat pattern

Assembly Functionality

- Interference analysis

- iMates—define, edit, suppress, delete, and use

- Constraints return associated entities

- Work features—create, query, and edit work planes, axes, and points

- Assembly Patterns—create, edit, and query linear and circular assembly patterns

- Ability to activate an occurrence for interactive edit

- Return assembly feature B-Rep information

Drawing Functionality

- Create, edit, and delete of
  - Standard drawing views
  - Projected drawing views
  - Weldments and weld prep views
  - Sheet metal folded or flat views

User Interface Customization

- Summary
  - "In the window" applications can modify user interface while maintaining ease of use
  - Users can personalize their Autodesk Inventor UI to increase productivity, including creating command buttons for user-created macros
- Create toolbars
- Add Context menu commands
- Many enhancements to User Interaction
- More control over client graphics text, including font, bold, italics, underline, and so on
- Environments—an application can create and activate an environment to streamline the UI to facilitate the tasks you need to accomplish; it sets the visible toolbars, context menu commands, menu commands, and so on (examples of environments are sketch and sheet metal)
- Document subtypes—"brand" a document that allows the application to set the environment when editing a subtype document

General Functionality

- Apprentice Server can display parts, assemblies, and drawings in another application using the native rendering engine of Autodesk Inventor
- DWF data is now generated dynamically from drawing (IDW) documents rather than displaying stale cached data
- Support the use of VBA functions within parameter expressions
- Access to the VBA project and functions and the ability to run them
- Support for new tolerance information associated with parameters
- OnFileResolution event enhanced to allow setting filename if Autodesk Inventor fails to resolve
- Better performance when using nested transactions
- Control over Application window and view windows
- Ability to save current view as a bitmap with specified resolution

User interface enhancements

UI Customization

Autodesk Inventor software allows you to tailor the toolbars to your liking. This can improve productivity by making frequently used commands easier to get to. You can now use the following enhancements:

- Place an iFeature on a toolbar button
- Place a VBA macros on a toolbar button

- Change toolbar/panelbar layout
- Create unique toolbars

Zoom at cursor

When using a wheel mouse, the Zoom operation is now centered on the cursor location.

Benefit: Users moving from AutoCAD will feel more familiar with Autodesk Inventor as the navigation tools now perform exactly the same. This familiarity speeds learning and helps ensure that AutoCAD users are productive on day one.

Background Images

Each color style now has three background options: one color, two colors, or image. One color is the default, and the graphics background is one solid field of color. Two colors allows for a gradient background that displays the background as a smooth blend from one color to another. Image, which is new in Autodesk Inventor 6, allows for a bitmap to be displayed as the graphics background.

Benefit: When creating images for presentations or marketing material it is often desirable to display a background image. Now Autodesk Inventor allows for unique and differentiating product images.

Draft Display Style
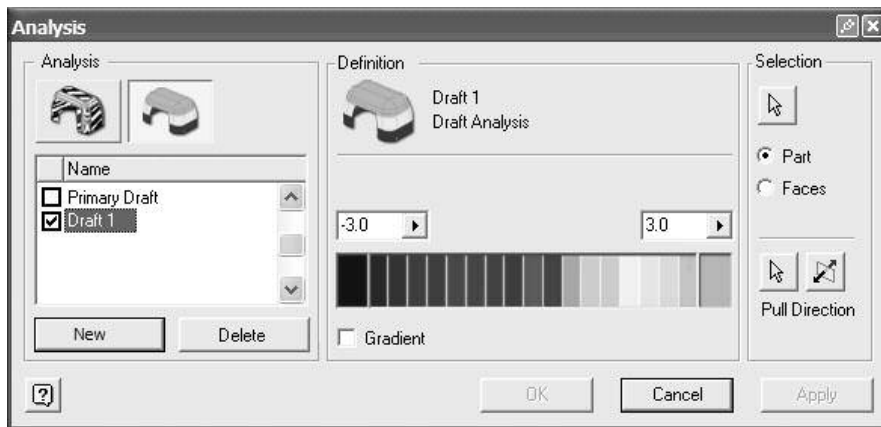
By providing a pull direction (vector on which the core and cavity separate) and the draft range, visual feedback is displayed on the model that helps analyze the manufacturability of the part. You may also create several definitions that allow you to review alternatives very rapidly.

Benefit: This type of analysis is primarily used to evaluate the manufacturability of molded parts and determine an optimal pull direction. Using your knowledge of the required draft (based on material and surface texture), you may analyze the part for probable areas of marring when being ejected from the mold cavity.

Zebra Display Style
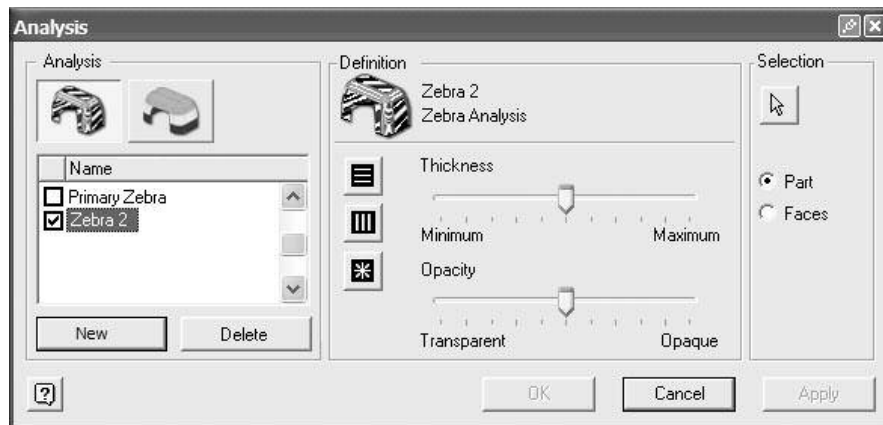
Zebra stripes are created by projecting infinitely long parallel lines onto the model. The appearance of the lines as they cross edges displays the level of continuity between adjoining surfaces.

Benefit: This form of display analysis allows you to more easily visualize the transitions from surface to surface to create a more aesthetically pleasing shape.

iProperties

Autodesk Inventor properties have been separated from traditional Microsoft® Windows® properties. They are now located in a separate section of the context menu in the Windows Explorer. By using these new properties, files can be exchanged between countries easier. Whether moving from Kanji to English to Russian or other languages, Autodesk Inventor file iProperties will continue to maintain the correct contents.

Benefit: You can be confident in the integrity of file properties no matter which language the file may be used in. File properties are often used in drawing title blocks and other annotations. Knowing that the contents of these properties are safe, you know the drawings that use these properties are also safe.

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