



## Features and Benefits

# What's New in Autodesk Inventor® 6

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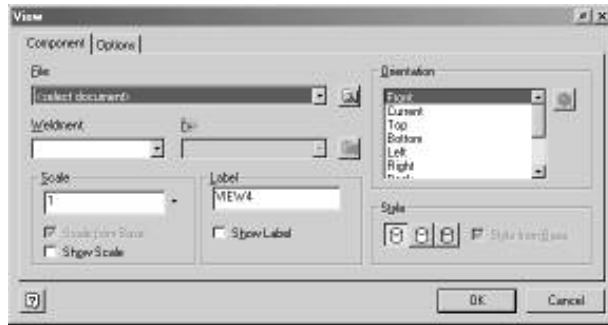
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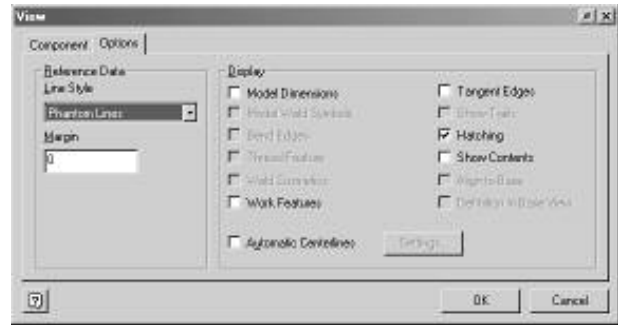
## Drawings

### Create View dialog box improvements

The Create View and Edit View dialog boxes are combined to offer more options during view creation. Access to view labels, tangent edge, bend line visibility, and automatic centerlines are available during view creation. New choices for making views of weldments offer control over the display of welded assemblies and the automatic display of weld annotations and symbols.



Component tab



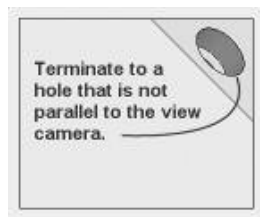
Options tab

**Benefit:** By offering more control at creation the new View dialog box provides greater flexibility in defining a view, ensuring that drawing layout times are greatly reduced. Views can be created automatically with dimensions displayed, centerlines applied, and special annotations visible.

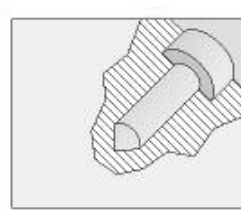
### Broken-out Views

Broken-out Views are a special view type that allows a drawing view to quickly reveal internal details of a design. Creating these views requires a boundary and a termination type. Boundaries are easily created by using a sketch in the drawing view. These sketches can be dimensioned and constrained to the view geometry, which allows them to update correctly should the size of the design change. The termination options for the broken-out section are as follows:

- To Sketch—a sketch drawn on a view that is a projection of the view having the breakout applied.
- From Point—specified distance from a point in the model view. The point may be specified in the base view or a projected view of the same part.
- To Hole—specify a hole feature whose axis provides the termination depth. This option works with holes at any angle.



Specify a hole as breakout termination



Hole axis becomes breakout depth

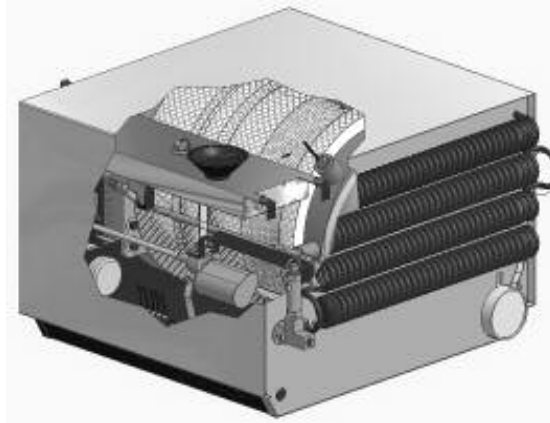
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- **Penetrate**—select the components that will have content removed from inside the break profile. This is especially useful for assembly views where you want to reveal an internal component or feature. Parts may be selected graphically or in the browser.

Use the Show Hidden Edges option to temporarily display internal details in the same view. This enables use of a single view to define a broken-out section.



Depth set to "Through Part"

In the above example, the outside cover and cooling coils were selected as Penetrate. The broken-out section affects only those two parts, cutting through them. The result is a window into the assembly.

Once a Broken-out View is created, an entry for the view is created in the browser. This makes it easy to locate and edit the view at a later date.

**Benefit:** Common in European standards, broken-out views allow drawings to reveal internal details so that they may be dimensioned directly on a main view. This removes the need to create a separate detail view or section view. Final drawing results are often clearer and less cluttered. In addition, broken-out views provide better support for drafting standards that require special hole dimension schemes based on broken-out views.

### Defer Drawing Update

Automatic drawing updates can now be turned off. This places the drawing in a deferred state. When a drawing is in this state, the drawing does not update to changes in the referenced part, assembly, or presentation. To help you recognize drawings in this state, the browser icon of the drawing changes to blue-green.

When using Autodesk Inventor rendered drawing views, options now control the quality of the display of the image that is cached to be displayed if Defer Updates is turned on.

Turning on Defer Updates and controlling cached drawing view quality is done using the Document Settings dialog box on the Drawing tab.



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### Defer Update document setting



### Image quality settings

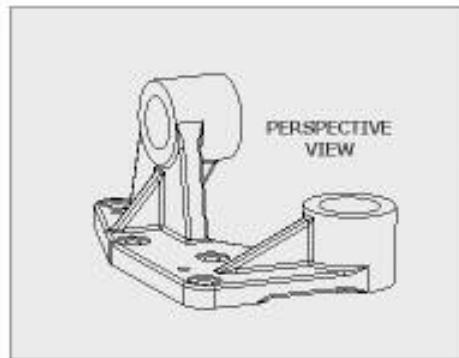
Most annotating can continue while the drawing is in the deferred state. This is similar to the drawing operations available when a model is not present. Note: annotation and drawing operations dependent on model features or other detailed model information are suspended when the drawing is deferred.

**Benefit:** Deferring automatic updates provides a simple way to archive a drawing. Drawings can be put into a deferred state until you specifically allow them to update. Deferred drawings are not affected by operations such as driving assembly constraints. Modifying a part, assembly, or presentation does not change the drawing. When the defer update state is turned off, the drawing views reacquire the model and update accordingly.

When working with very large assemblies it is often easier to wait to process view updates for several changes at once instead of doing them incrementally. Using Defer Update, these drawings can be updated only when and if needed, saving time and improving performance.

### Perspective drawing views

Perspective views are available in drawings. Balloons, leader text, user-defined symbols, and drawing weld notes are supported in perspective views. The primary benefit is using the view for pictorial purposes.



Perspective drawing view

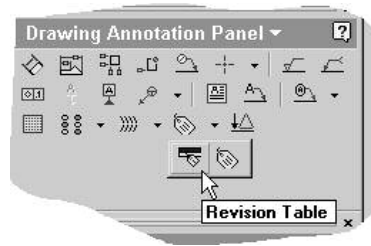
### Revision block

A Revision block command is provided in the Annotations Panel toolbar. Activating the command provides a revision block attached to the cursor for placement in the drawing. One revision block per drawing is honored. If more than one is used in a drawing, all instances reflect the same information.

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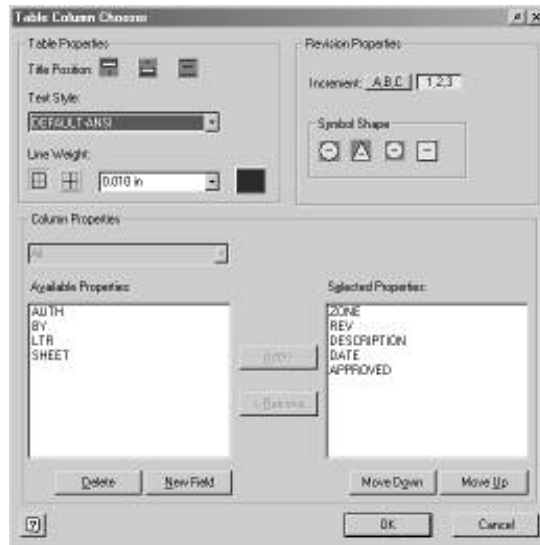
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Drawing Annotation Panel  
toolbar, Revision block

Various formatting options are available, such as title position, text style, line weight, line color, type of increment (alphabetic or numeric), and symbol shape. A set of default property columns is provided with the ability to add user-defined columns.



Revision Block Edit dialog box

A diagram showing a 'Revision Block' and a 'Revision Tag' in a drawing context. The Revision Block is a table with 5 columns and 3 rows. The first row contains specific revision data, and the second row contains headers. The Revision Tag is a small symbol to the right of the first row of the table.

REVISION HISTORY				
ZONE	REV	DESCRIPTION	DATE	APPROVED
A3/A4	1	ADD REV BLK	06/04/2002	A. USER

Revision Block and Revision Tag example

Place the revision block, then right-click to access its formatting options. In the drawing, double-click on the text within the revision block and edit as needed to provide revision information. The revision block border and lines are graphically editable in place. Rows can be added via the context menu.

Complementing the revision block is the Revision Tag. The command accompanies the Revision block in the Annotations Panel toolbar and provides a tag formatted according to the options selected in the Revision Block dialog box. Revision numbers/letters increment automatically.

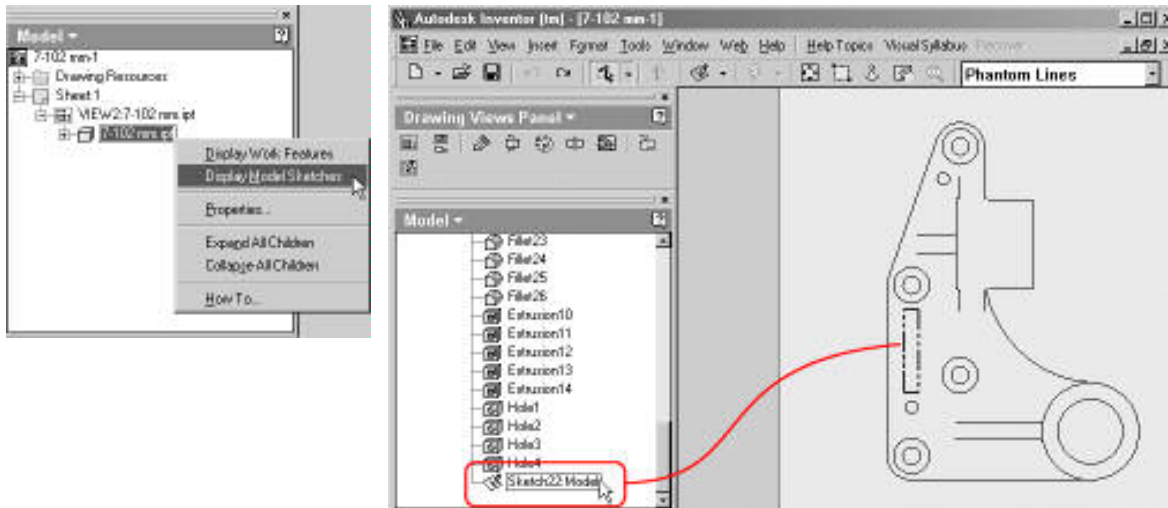
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**Benefit:** Revision block and tag annotations saves the work of constructing a revision block manually and provides easy drawing revisions.

### Show Sketches in Drawing Views

Model sketches can be displayed within the context of the model view. For example, if you sketch a silkscreen layout on a part face, you can use that unconsumed sketch in the drawing to associatively document the silkscreen layout.

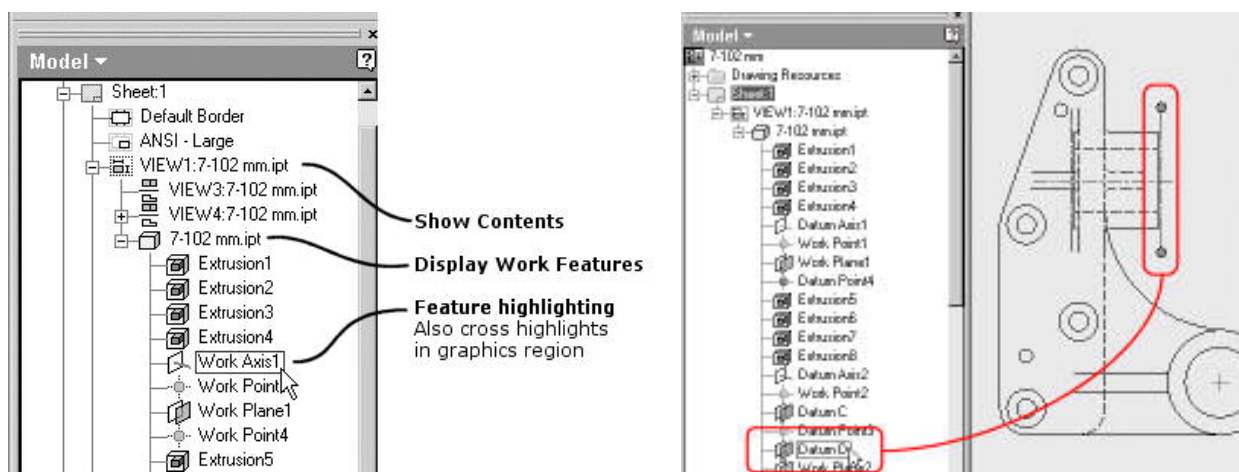


### Show and reference work features in drawing

Work features are often used as datums when constructing models. Model work features can now be displayed in the drawing, and any annotations to them are associative.

Work features have their own line style, allowing you to define how they appear in the drawing. These are maintained along with the other line styles in the Standards dialog box.

To display model work features, right-click on the view and choose Show Contents. Next, right-click on the desired component in the browser and select Display Work Features. The browser is populated with the model's visible work features. These work features have a visibility toggle so you can specify what is visible in the drawing. Only visible model work features populate the drawing browser.



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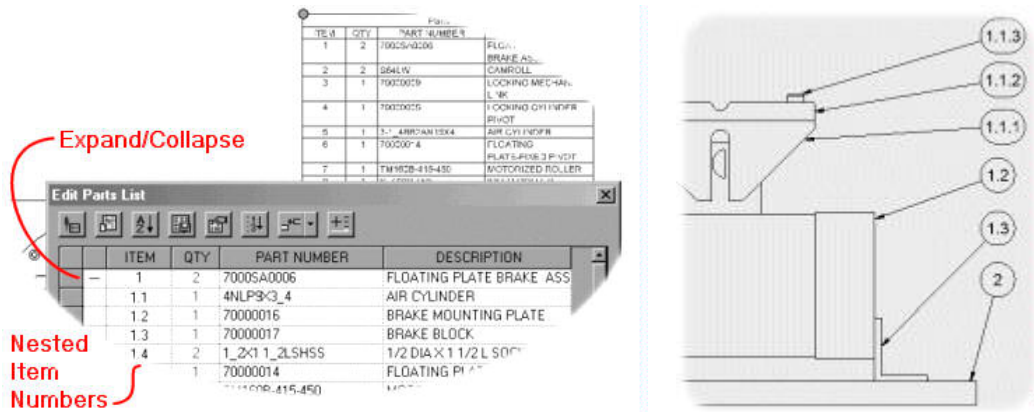


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**Benefit:** Work features used as datums are a common occurrence in part and assembly models. Using these in drawings ensures that model datums are honored in the documentation. Annotation is simple and associative.

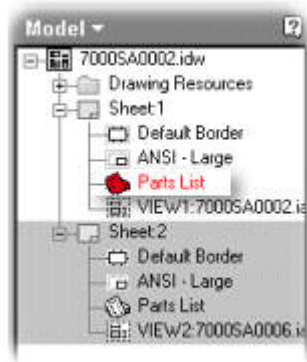
### Parts Lists

Autodesk Inventor 6 provides selective expansions of nested parts lists. Assemblies containing subassemblies present a "+" sign in front of the item number in the Parts List dialog box. The expansion occurs when the plus sign is clicked, contraction when the minus sign is clicked.



The first level of nesting displays as 1.1, 1.2, 1.3, . . . and if item 1.3 is a subassembly to be expanded, the second level of expansion is 1.3.1, 1.3.2, 1.3.3, . . . Balloons can display nested parts list item numbers.

When a parts list becomes stale, needing an update, the browser node associated to the parts list turns red as a visual cue.



Parts list needing update

Right click on the browser node and select Update to freshen the entire parts list, or use the Parts List edit dialog box to update discrete fields of the parts list.

Freezing and thawing of individual cells in the parts list control is accessed through the context menu of the selected cell. Frozen cells do not respond to edits or updates, and their state can be switched at any time.

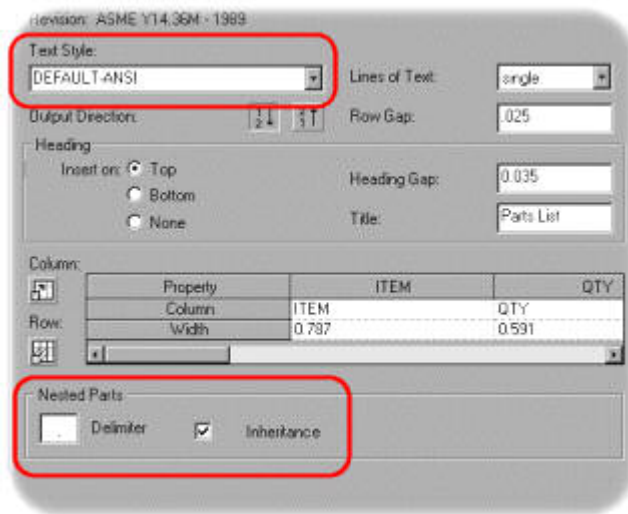
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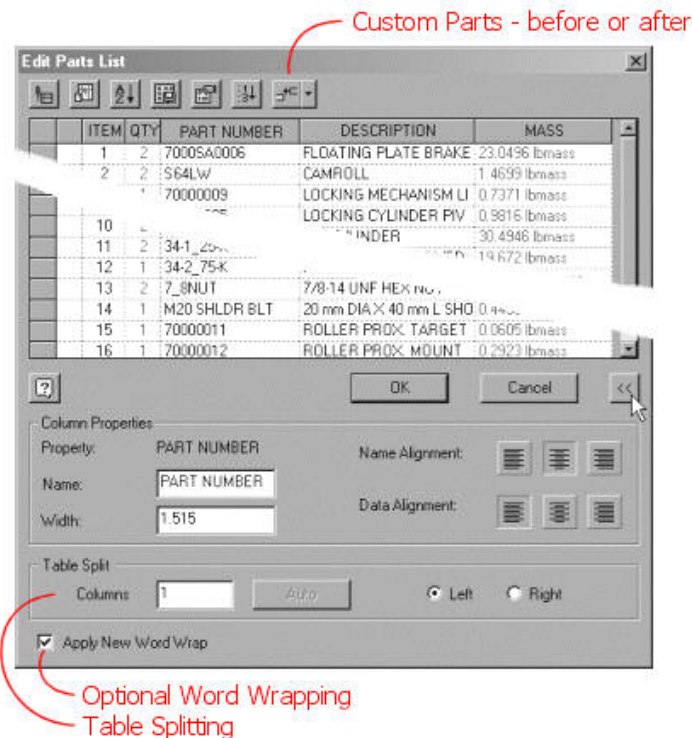
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The Parts List tab in the Standards dialog box includes new controls for specifying a Text Style, nested list delimiter (any character may be used), and an inheritance option. Inheritance propagates changes in parent parts to nested child parts.



New Parts List options

Additional parts list options include custom parts with a before/after selection. Word wrapping is optional, and table splitting is available.



New options for editing a parts list's formatting

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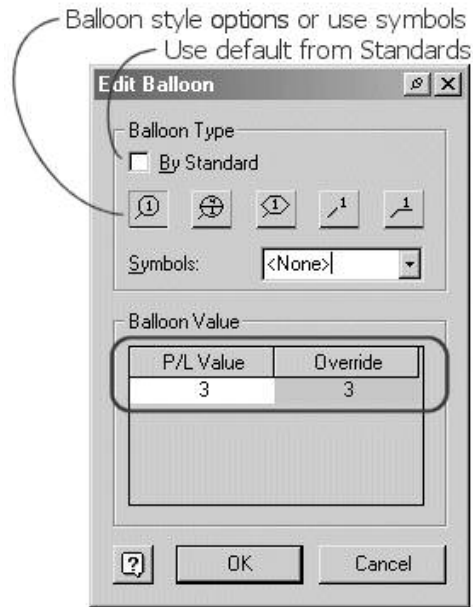
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## Balloons

Balloons are now easier to use and offer more editing and formatting options.

The Balloon tab in the Standards dialog box has options for specifying the balloon text style, balloon type, and an offset value used in conjunction with balloon alignment commands.

Improvements in balloon editing and formatting options include a dialog box for editing individual balloons. Double-click or choose Edit from the context menu to display the Edit Balloon dialog box. By default, formatting is set to that which is defined by the drawing standard. If special formatting is required for a particular balloon, make a selection from the other available balloon styles.



New Balloon editing options

Additionally, several other new balloon options are possible.

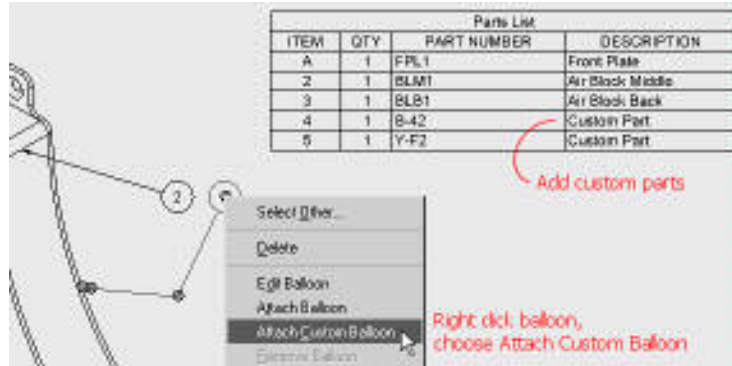
- User-defined symbols can serve as balloons.
- When overriding balloon values, two balloon values are displayed, one for the default parts list number, the other for a user-defined override. Custom override values directly affect what is displayed in the balloon instance. For attached balloons, all members of the balloon set are presented in the dialog box.
- Balloon leaders now have orthogonal snapping similar to other annotations.
- When balloon override content exceeds the balloon size, the balloon becomes obround to fit the content.
- Auto Balloon supports Part filtering and multiple select via Ctrl+Click for ballooning discrete selections. Make selections first, and then invoke the Auto Balloon command.
- Horizontal and vertical alignment supports multiple balloon selections. The first object in the select set serves as the anchor for the other selections. Offsets enable specific value.

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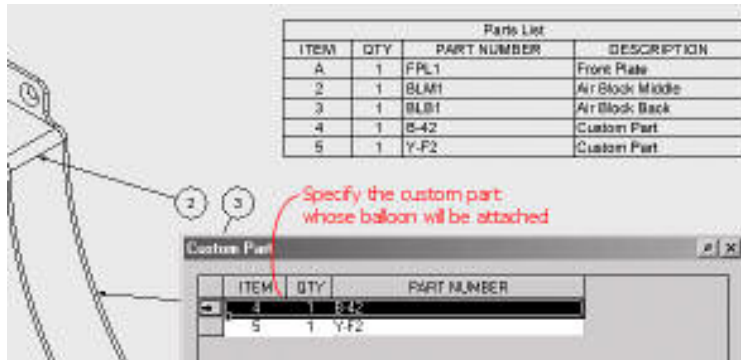
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- Support for attaching custom balloons to existing balloons has been added. To make use of this feature, add custom parts to the parts list. Then right-click on a balloon you want to attach the custom part to, then choose Attach Custom Balloon.



A list of custom parts is presented. Specify the custom part whose balloon will be attached to the selected balloon.



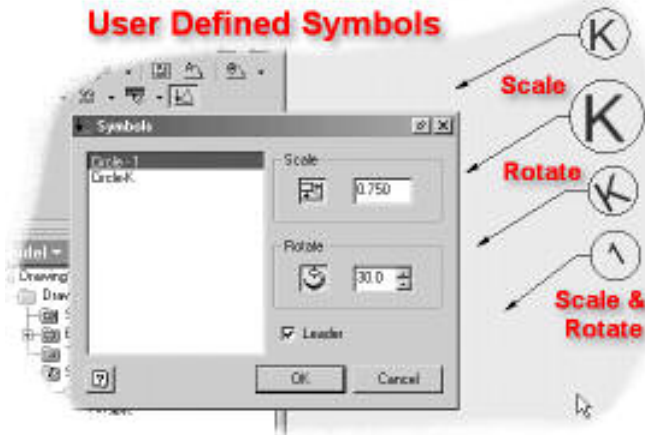
Position attached balloon and the task is complete.

## User-defined symbols

User-defined symbols are now available from the Annotations Panel toolbar. The Symbol command dialog box presents a list of symbols in the drawing, options for scaling and rotating, and whether or not a leader is used with initial placement. Insertion points are optional when creating symbols. These are used for placement, scaling, and rotation. When an insertion point is not used, the operations occur relative to the geometric center of the symbol.

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New options for editing user-defined symbols

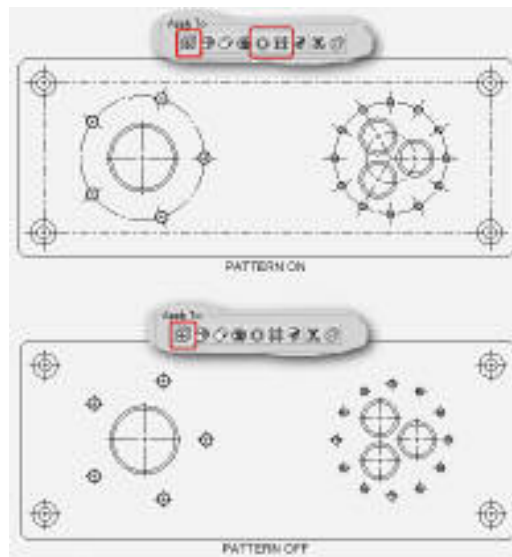
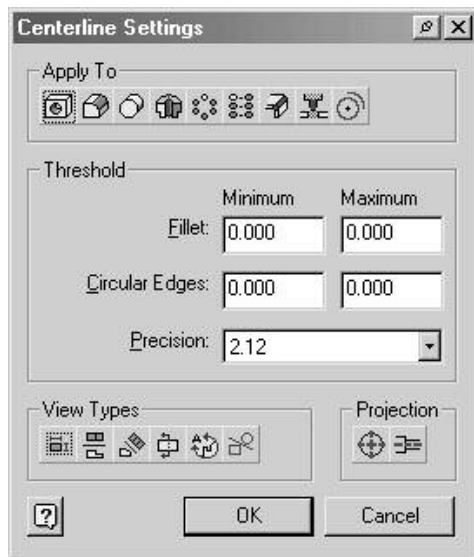
Previous methods (click-and continue or double-click) for placing leaderless symbols continue to be available (when the leader option is checked). If a leaderless symbol is placed while inside a view boundary, it associates with the view.

**Benefit:** Symbols are located in an easy to access list. They scale and rotate about the insertion point (if used) or about the geometric center of the symbol.

### Automatic centerlines and centermarks

Automatic centerlines and centermarks are now available when creating or editing views on a drawing. A drawing document can be set to create automatic centerlines and centermarks, or they can be applied per view. Access the command and settings in the Document Settings or the View Create/Edit dialog box.

Specify the feature types, view types, and projection types to receive centerlines. There are minimum and maximum tolerance values for fillets and circular edges. The tolerances allow you to exclude specific ranges of circles and fillet radii. Precision does affect the tolerances.



Supported Features

Supported Views

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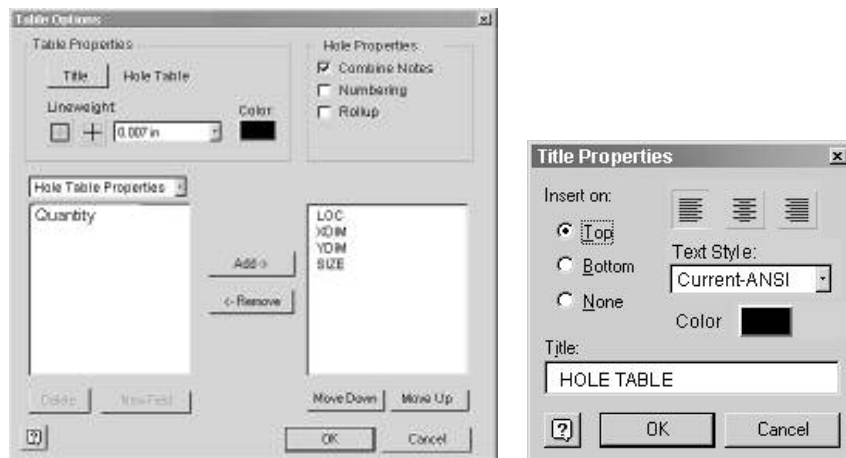
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Hole	Independent and base
Fillet	Projected
Extrude—whole or partial cylindrical only	Auxiliary
Revolution	Section
Circular Patterned	Detail
Rectangular Patterned	Drawing Sketches (includes draft view sketches)
Bend (Sheet Metal)	
Punch (Center of punch)	
Sketch geometry—arcs and circles	

**Benefit:** Centerlines and centermarks are added to drawings automatically, reducing the time-consuming task of locating and placing them. Discrete feature and view options give greater control over the objects and views receiving centerlines and centermarks. Centermark extension lines can be lengthened or shortened. The centerline now uses the current line style from the standards.

### Hole table enhancements

Autodesk Inventor 6 has new formatting options for hole note tables. Hole tables now have an entry in the browser. The new formatting options for hole tables include text style, line weight, color, column chooser, and user-defined columns.



Click on an item in the column chooser to access formatting options for that column

Select a row in the hole table, right-click, and in the context menu choose Split table. The hole table splits at the selected location and generates a new table with the remaining information. Multiple splits are supported.

The Combine Notes feature consolidates the hole table by combining like holes into one notation in the table.

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Hole Table			
LOC	XDIM	YDIM	SIZE
A1	0.45	0.34	1/4-20 UNC - 1B ✓ Ø0.38 X 82"
A2	1.44	1.19	
A3	0.45	1.19	
A4	1.44	0.34	
B1	0.77	0.84	Ø0.25 ± 0.75

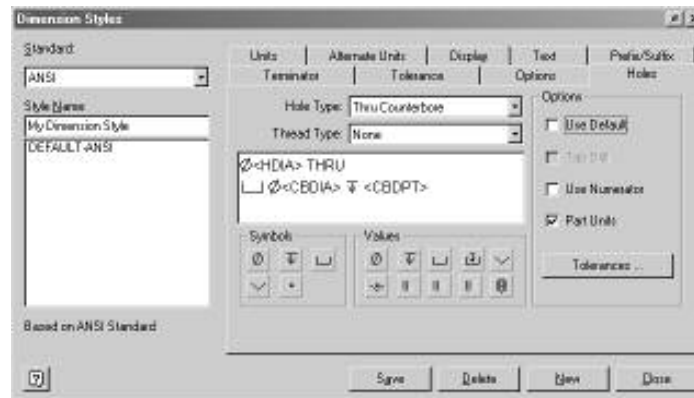
Hole table with notes combined

The Numbering option changes the table location tags and hole tags from alphabetical to numerical. The Rollup option forces like holes to be combined and quantity listed. The letters or numbers update and re-sequence. The By Type option makes it possible to select holes of one type and create a separate table for that selection.

### Hole notes

Drawings now allow editing and defining of custom hole notes. Controls for hole note content, tolerances, text, and symbology are all provided and can be customized as needed.

Hole notes are made of a mix of user-defined words and symbols along with special “tokens” that are placeholders for the specific hole information extracted from the 3D design. The hole note dimension tokens can be arranged and formatted according to specific needs. Options include tap drill information, numerator (for hole counts on patterned holes), an option to use the part units or drawing units, and tolerance settings.



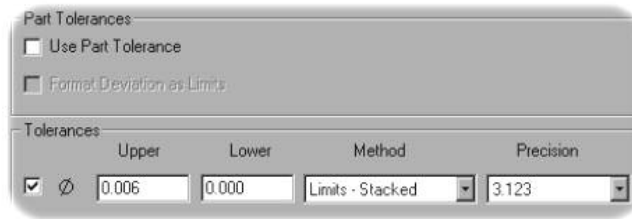
Hole notes tab

Part tolerances, also new in Autodesk Inventor 6, may be used, or drawing tolerance values can be specified.

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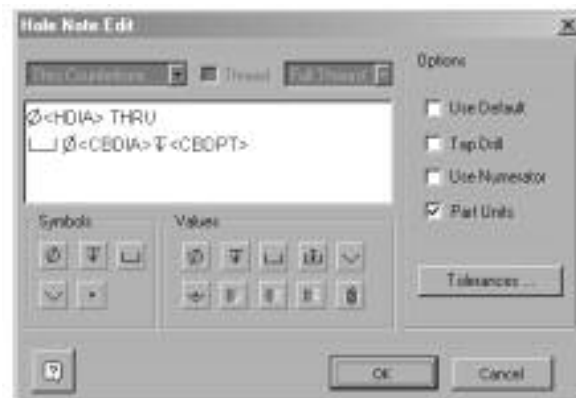


Part Tolerances dialog box



Hole note with numerator

Modifying hole notes is simple. Choose Edit Hole Note in the context menu and the Hole Note Edit dialog box appears. Hole notes can be overridden individually.



Hole Note Edit dialog box

**Benefit:** Drawings with hole notes can be made to better match your company's drafting standard in less time and effort.

### Text Improvements—Alignment

When working with blocks of text for notes in a drawing, Autodesk Inventor 6 now supports alignment of General Notes, Sketch-Text, and Parameter-Text. When two or more text objects are selected, the context menu includes the Align Text command. Selecting this command aligns all selected text with the position of the first object selected.

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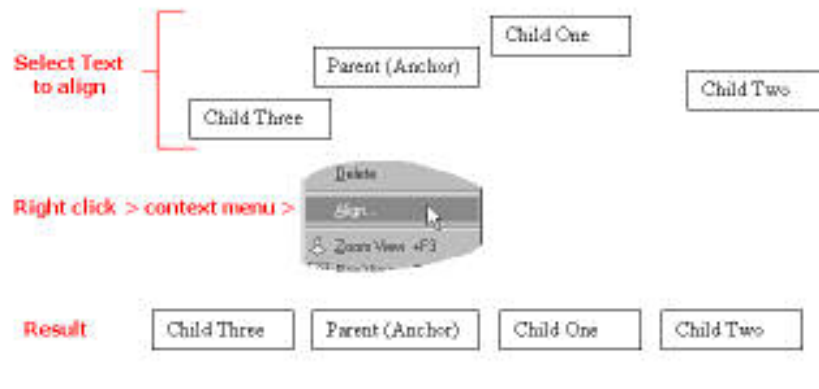
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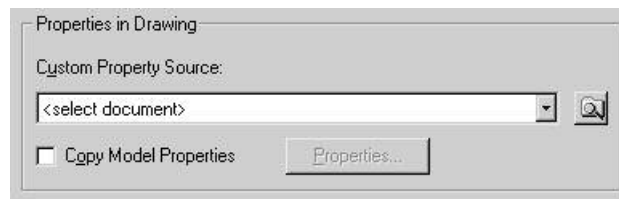
Offset value is the minimum distance between the text bounding boxes



**Benefit:** Notes can be difficult to precisely align and distribute. Autodesk Inventor 6 makes this a simple and easy task.

### Use part/assembly model properties in IDW

There is new functionality for use of properties. In the Document Settings > Drawing tab, you can specify a Custom Property Source. The custom property source can be any Autodesk Inventor file or template. The custom properties in that file are made available for use in the drawing through the Copy Model Properties checkbox.



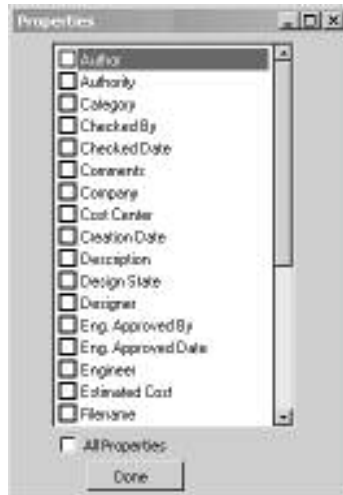
File properties source chooser

Checking the Copy Model Properties option enables the Properties button. Clicking on this button displays a dialog box with the list of all model properties available to the drawing. If a Custom Property Source has been specified, the list includes all custom properties from that source. Otherwise, the list contains only the standard model properties.

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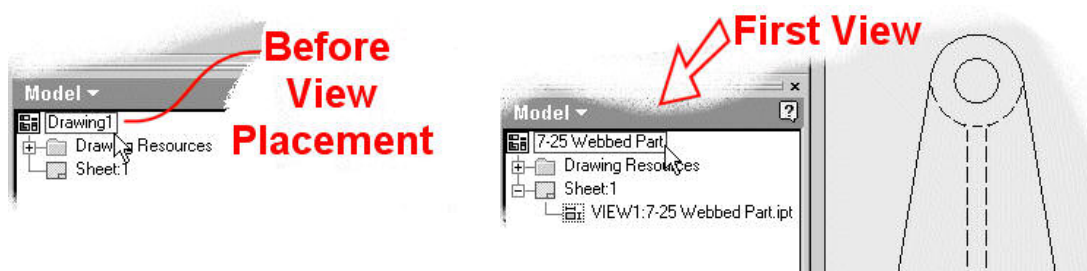
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Properties chooser

The default drawing filename is now derived from the model's filename. The first model view placed provides the name. When the drawing is saved for the first time, this name is presented as the suggested filename.

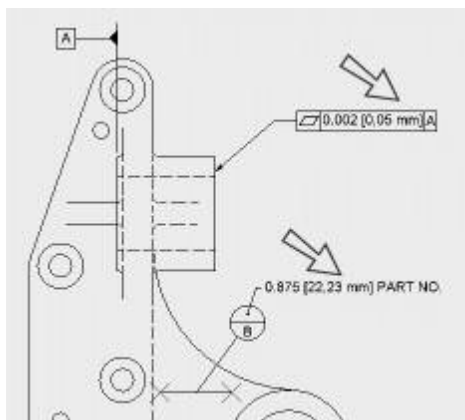


New drawing filename

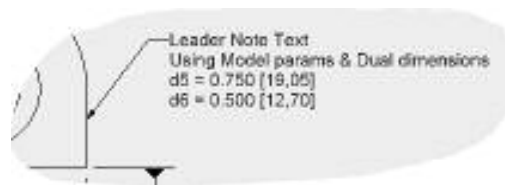
After first model view

## Dual Dimensions

Feature control frames, datum targets, and leader text now have dual dimension reporting. Each object has separate unit controls to allow greater flexibility in drawing content.



Feature control frames and datum targets



Leader text

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## Dimension Styles

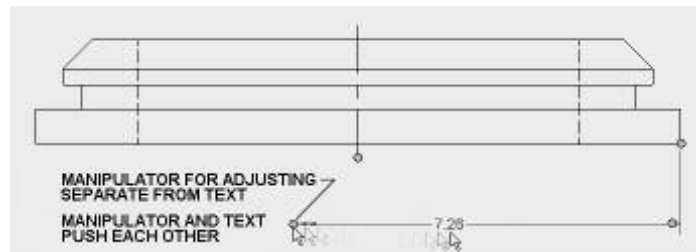
You can select several dimensions of different styles and from the context menu create a new style applying it to the selected dimensions upon completing style definition.

The style of the first dimension in the selection set is used as a basis for the new style definition. The style, once defined and the dialog exited with OK, is applied to all dimensions in the selection set.

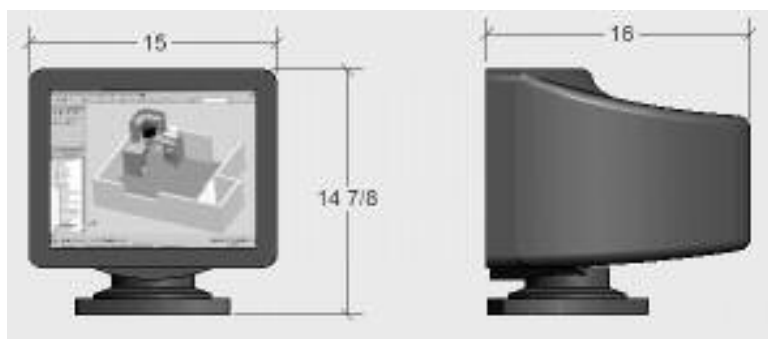
Tolerances can use Text Styles (Dimension Style > Tolerance tab). Dimension tolerances can be defined as part of the dimension style. Dimension tolerances can now use a different text style from the dimension text. There are three justification options and a size option. The size option is available when the text style is set to none.



Linear Symmetric Arrowhead—you are now able to control the position of the arrowhead separately from the text. This improves drawing cleanup.

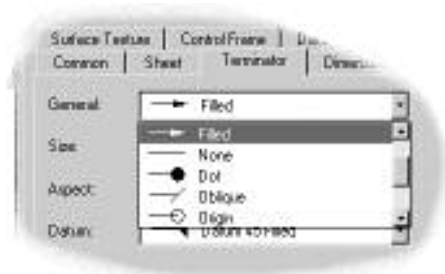


Terminator List—the full list of terminators is available wherever terminators are defined or edited. For example, you can define a dimension style that uses the oblique terminator exclusively.

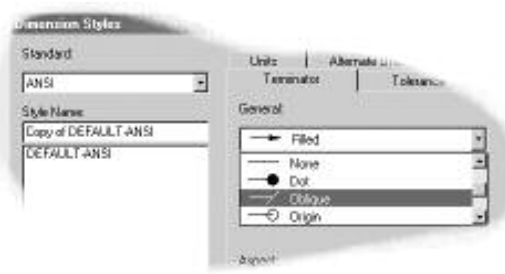


Oblique terminator in a dimension style

## Features and Benefits: What's New in Autodesk Inventor 6



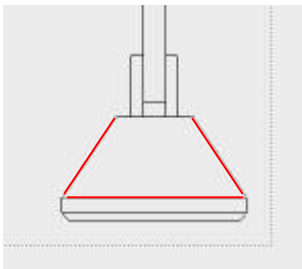
Standards dialog box



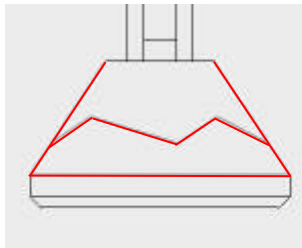
Dimension Style dialog box

### Project View Edges to Sketch

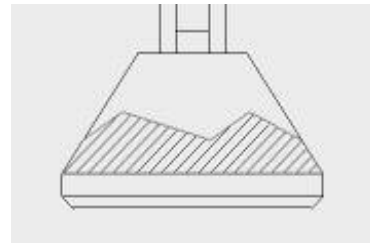
When drawing a sketch associated to a view in a drawing, you can now project the edges that are needed for constraints, sketch elements, or other uses directly. Projecting the edges creates a reference sketch element that is associative to the originating view element.



Start a sketch associated to a view.



Sketch, using project edge as needed.



Complete the task by adding hatch or fill, if required.

**Benefit:** Easily create bounded areas for hatch or fill. Projected edges are associative, so when the model changes the edges will update.

### Weld Features support in Drawing

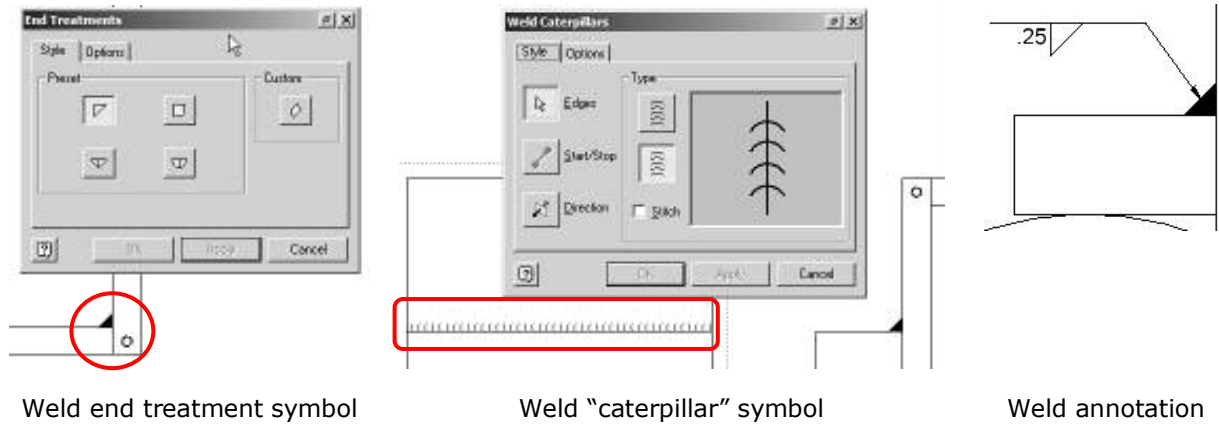
When creating drawings of a weldment, special symbols like caterpillars and end treatments are now available making annotating a weldment design fast and easy. Autodesk Inventor 6 offers new support for

- Weld annotations—the weld symbol calling out the details required to produce the weld
- Weld bead—the material created by welding
- Weld symbol—a simplified representation of the weld bead used to illustrate the weld bead in a drawing. Often these are small polygons, mostly triangles, that mark the edge direction and size of the bead for clarity.

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## Features and Benefits: What's New in Autodesk Inventor 6



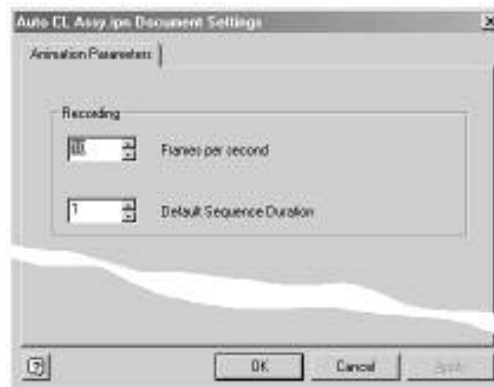
Weld symbols can be created manually using the new weld annotation commands. If solid weld beads were created in the designs, the weld symbols can be created automatically at time of view creation. For both solid and cosmetic weld bead definitions, the weld annotation can be retrieved. The context menu of the view presents the options to display symbols and annotations as they apply.

**Benefit:** Weldment drawings are specialized drawings that communicate the size, type, and manufacturing process of welds. Providing a complete set of tools to document weldments quickly allows you to create complete detailed drawings needed to manufacture welded components.

## Presentations

### Animation Recording

There are two new settings for presentations. In the Tools > Document Settings dialog box, there are settings provided for adjusting the recording timeframe and sequence duration.



The number of frames per second for recording speed can be set and the length in frames derived from the two settings. For example, if the presentation is set to record at 10 frames per second, the sequence duration is set to 1 (second), assuming all sequences have the same duration. The presentation animation contains 30 sequences of 1 second each. Therefore the animation will be 30 seconds long and consist of 300 frames.

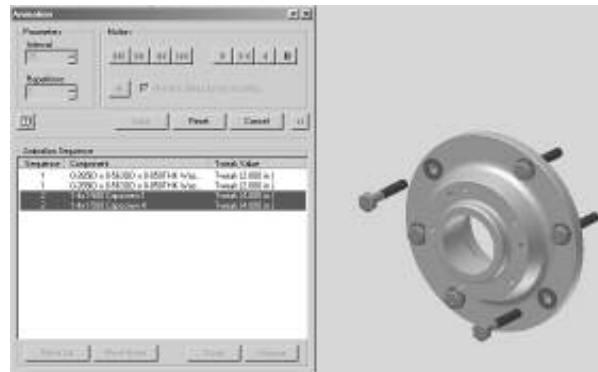
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The default length of sequences can be set. This applies to all sequences created after the setting has been applied. This provides the option to establish standard settings for animations.

### Playback Highlighting for Sequences

Sequences are displayed in the More section of the animation dialog box. When the animation is played while the More section is displayed, each sequence is highlighted in the dialog box as it is played back in the window.



**Benefit:** Whether you're editing or viewing a presentation, the highlighting makes it easier to match sequence names and settings with the animation window.

## Part Modeling

### Unified Shape Technology

Unified shape technology is an exciting new capability provided by advances in Autodesk Shape Manager, the solids modeling kernel used in Autodesk's products. Unified shape technology specifically describes the new hybrid modeling tools in Autodesk Inventor.

Hybrid modeling tools allow both traditional surface modeling and advanced parametric modeling to be used seamlessly together. Autodesk Inventor 6 designs can be converted from solid to surface at any time and maintain their parametric history for fast and easy design changes. This hybrid approach allows complex shapes to be tackled by building rough solid shapes that are then sculpted using surface tools.

Surface creation tools include extrude, loft, revolve, sweep, thicken/offset, promote, and derive.

Hybrid tools include

- Knit—can convert surfaces to solids
- Delete face—can convert solids to surfaces
- Promote—can promote surfaces from one part to another
- Derive—can derive surfaces from one part to another

**Benefits:** Autodesk Inventor offers true hybrid modeling tools to tackle complex shapes with simple-to-use tools.

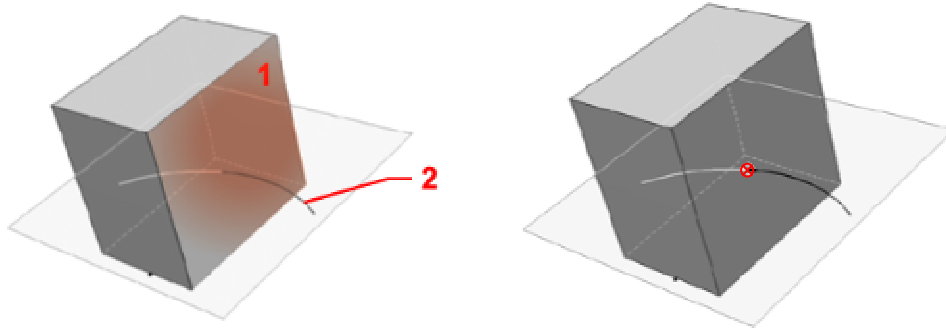
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## New Work geometry construction methods

When working with complex shapes, work geometry can be used to help define construction geometry thereby helping to break down the design task and make the definition of complex geometry easier. Autodesk Inventor 6 extends the construction methods available, covering a wider range of spline and complex shape problems.

Work point at physical intersection of face and sketch curve or model edge



Face/Sketch inputs

Results

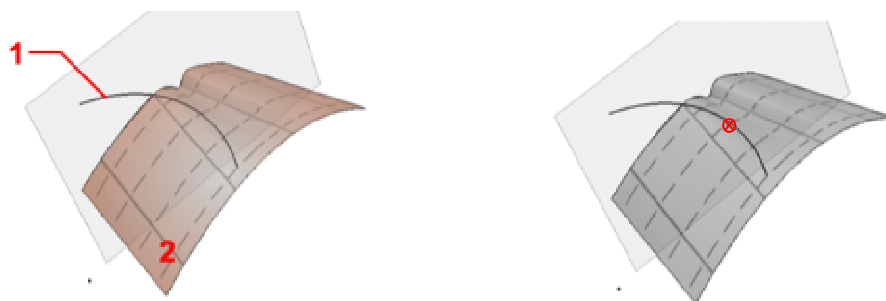
Work point at physical intersection of plane and 3D sketch, sketch curve or model edge



Plane/Sketch inputs

Results

Work point at physical intersection of surface and linear sketch segment or model edge



Surface/Sketch inputs

Results

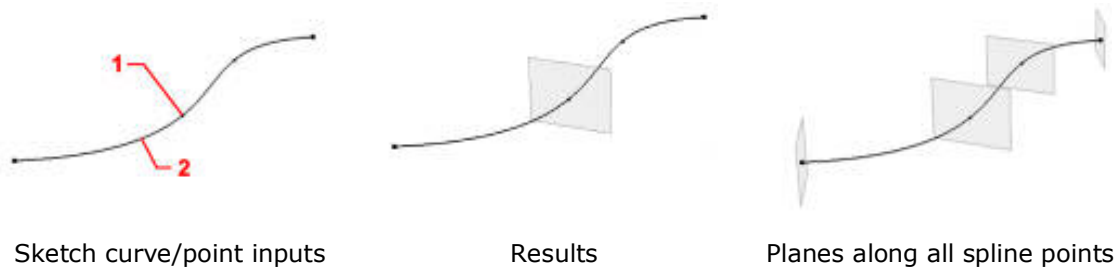
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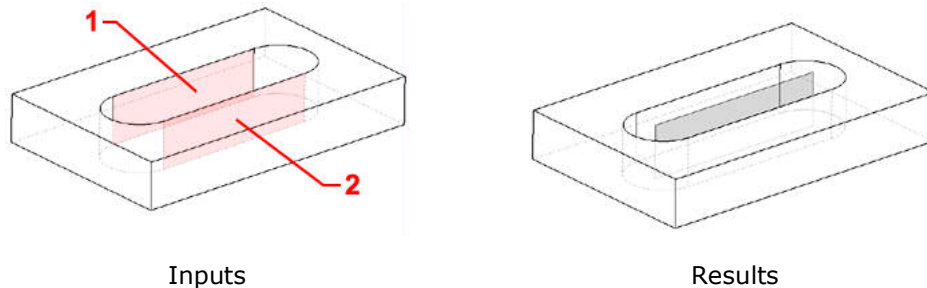


## Features and Benefits: What's New in Autodesk Inventor 6

Work plane normal to point on a sketch curve or model edge



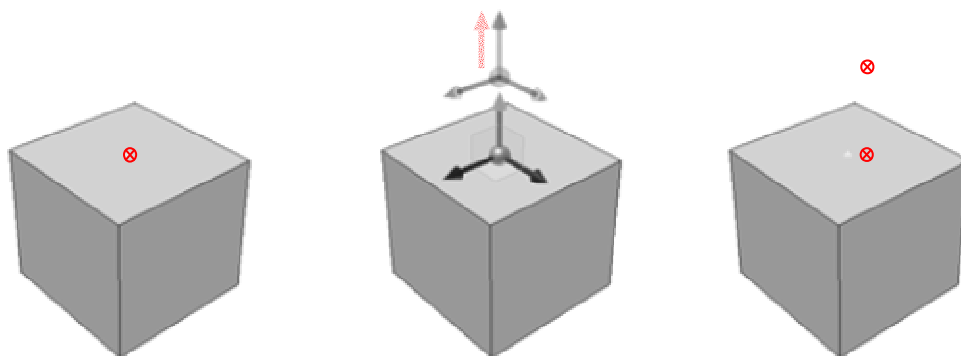
Work plane between parallel planes



**Benefit:** Consumer products, plastic parts, and castings require complex shapes. Construction geometry allows these designs to be accomplished by giving you tools to break down the geometry into simple features. These expanded construction techniques directly impact your ability to define complex designs.

### Grounded Work Points

A new grounded work point type is available that can be positioned in space using a precision 3D manipulator. These points can be used in 3D sketches and to provide a solution for design problems that require explicit, nonparametric points in space. The grounded work point command is located under the work point flyout in a toolbar or the panel bar.



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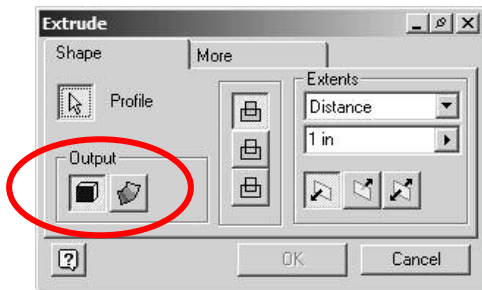
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## Features and Benefits: What's New in Autodesk Inventor 6

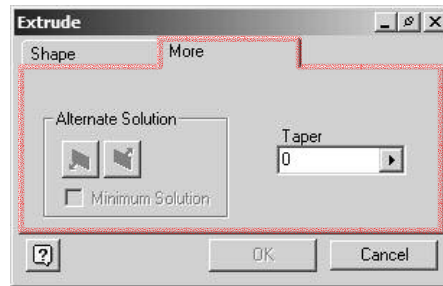
Start with a point or vertex    Using the point manipulator    Final points placed

### Extrude Dialog Box

The Extrude dialog box has been rearranged to present information more clearly. A More tab has been added to make working with options easier. The dialog box now can account for new features like termination options and open profiles.



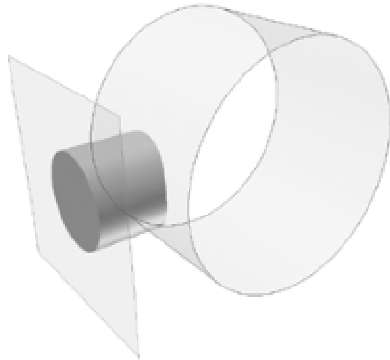
Extrude dialog box



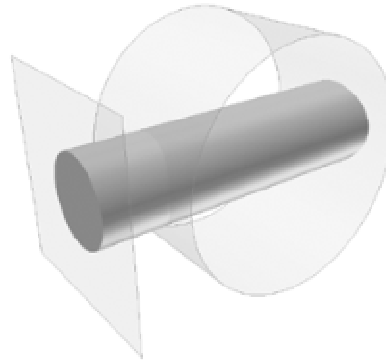
The More tab

### Extrude terminations

When extruding to closed loop faces or surfaces, you can now define a minimum or maximum termination solution. These options are available from the More tab in the Extrude dialog box. A new termination type has been added to the To Next option. This allows an extrude to terminate to any surface or solid body by specifying an additional terminator.



Minimum termination solution



Maximum termination solution

### Extrude Open Profiles

Open profiles are composed of sketch geometry that do not form closed loops. This technique uses existing model geometry to define closed profile regions. Open profiles offer advantages because model edges do not need to be projected into the sketch to create a closed loop. This allows the model to change topology without causing the open profile to fail. Models can be easily changed to explore more design ideas with fewer downstream failures.

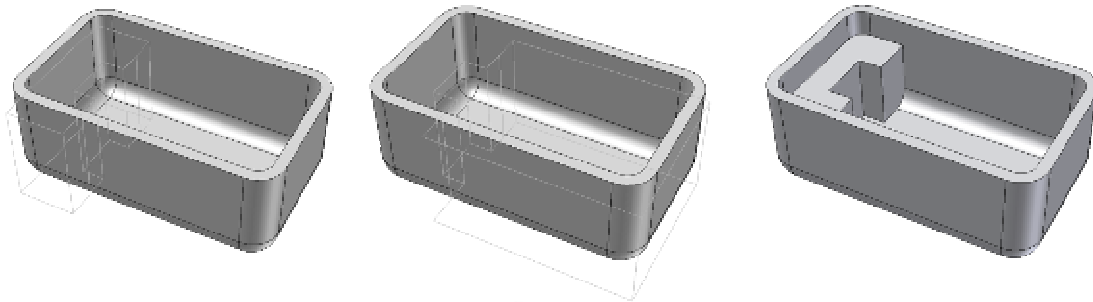
Open profiles were supported in rib since release 4, and are now allowed when creating extrudes. When using open profiles additional user input is needed as there are often

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## Features and Benefits: What's New in Autodesk Inventor 6

multiple solutions possible. This input is dynamic and uses dynamic previews to help you easily visualize the results.



Side A

Side B

Final fill in two directions

**Benefit:** Open profiles yield models that are significantly more change-tolerant. You are freed from worrying about making drastic design changes and having to clean up failed features. This simple solution allows you to explore more complex problems, resulting in more innovative designs.

### Loft

Significant changes have been made to the Loft dialog box to provide better control and definition of complex shapes. Using new Autodesk ShapeManager functions, Loft now allows multiple sections and guide curves. Section curves may be 2D sections, 3D sections, or model edges. Guide curves may be 2D sketches, 3D sketches, or model edges. Conditions allow for shape through profiles to be controlled without rails. Conditions can be tangent or an angle and can include a weight factor to increase or decrease the effect on shape.

Mapping of profile vertices from one profile to the next allows very precise control of topology change and twist minimization. Autodesk Inventor creates automatic point mapping that can be edited and changed should an alternative shape be desired.



Loft curves are used to select profiles and guide curves, or rails

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## Features and Benefits: What's New in Autodesk Inventor 6



Loft conditions allows control of how the shape passes through each profile  
(Used when no rails are selected)



Loft transitions for shape and twist control

**Benefit:** With increasing emphasis on aesthetics in design, even for industrial machines, Autodesk Inventor 6 introduces powerful new tools. Many new complex designs can be defined using the new Loft functions and sculpted with Autodesk Inventor 6's surface tools. You can also expand the markets you design for.

### Knit

In order to use surfaces for Replace Face or Split Part operations, you sometimes need to group surfaces together. These groups, called quilts in Autodesk Inventor, can be created using the Knit command. In addition, if the group of surfaces completely defines a closed volume, Autodesk Inventor can convert this volume into a solid on which to do further editing using other parametric features.

### Thicken Command

The Thicken command can be used to add material to any surface, face, quilt of surfaces, or selection of faces. The Thicken command is very similar to Extrude as it allows a direction vector to be selected. It also allows a join or subtract operation to be defined. Thicken is akin to shelling and exhibits many of the same behaviors.

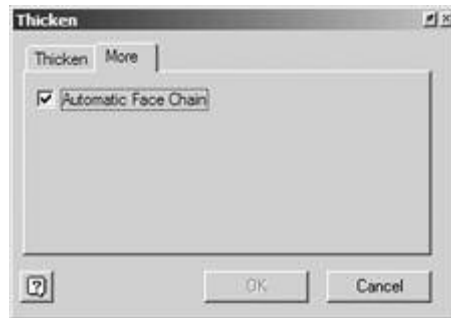
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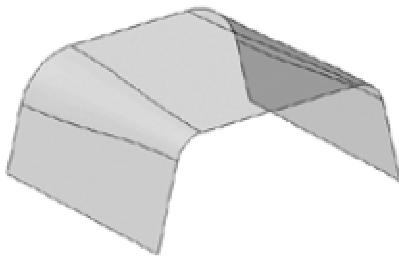
## Features and Benefits: What's New in Autodesk Inventor 6



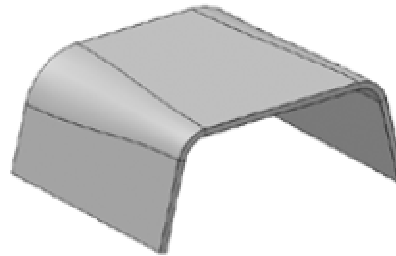
Thicken/Offset dialog box



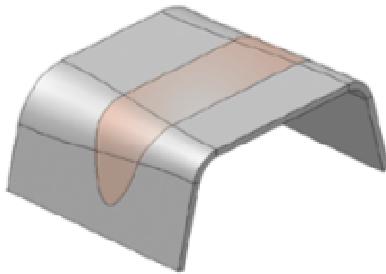
The More tab



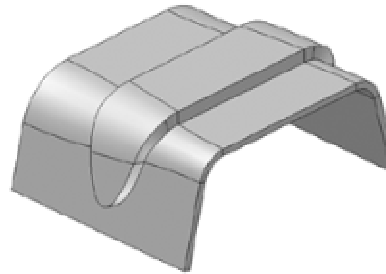
Surface



Thicken results



Solid



Thicken results

**Benefit:** Thicken offers alternatives to shelling to create thin-walled parts. Offset provides an often needed tool to create construction surfaces. Both these tools are common constructions when working with plastic parts or more aesthetic shapes.

### Replace Face

Replace Face allows any model face to be replaced by a surface or quilt. Replace Face allows solid designs to be sculpted with surfaces to achieve more aesthetic shapes common in plastic parts. Replace Face does not require that the new surface extend past existing model surfaces. Replace Face also cuts or extends the solid model depending on how the new surface intersects.

Replace Face dialog box

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