AUTODESK®
INVENTOR 11

Autodesk

Technical What's New

Contents

The Autodesk Inventor® product line is the best choice for AutoCAD® users. It provides a comprehensive, integrated set of design tools for manufacturing companies including Autodesk Inventor® Series for 3D design and documentation, Autodesk Inventor® Professional for creating routed systems and validating designs, AutoCAD® Mechanical for 2D drawing and detailing, and Autodesk® Vault for data management. The Autodesk Inventor product line delivers enhanced 3D productivity while preserving your 2D engineering designs through true DWG compatibility. Autodesk Inventor also delivers a new ground-breaking modeling paradigm, Functional Design. Functional Design allows designers to move beyond geometric modeling and into an environment where they can focus on the problem they are trying solve rather than spending time solely on the 3D geometry required to build the design.

Assembly Configurations

To support the design and documentation of product families, Autodesk Inventor 11 introduces support for assembly configurations based on iAssemblies. Similar in concept to the existing iPart tools, the iAssembly is a table-driven representation for a set of related assemblies.

By creating an iAssembly, the user is able to define a set of related assemblies simply by editing the iAssembly table. Each of the resulting member assemblies is a native Autodesk Inventor assembly, created on demand.

Assembly-level modifications include: replacing components, excluding components, adjusting constraint and parameter values, and adjusting BOM properties.

With the new Assembly Configurations you can:

- Create an iAssembly factory from which member assemblies can be created by editing the associated assembly table.

- Specify Part Number indexing and member naming schemes.

- Use Autocapture mode to modify the iAssembly table by capturing changes made while modeling.

- Use Microsoft Excel® [1] to incorporate logical expressions in the configuration table.

- Place a selected member of an assembly configuration into another assembly.

- Specify an iAssembly member in consuming documents such as drawings, presentations, and derived assemblies.

- Add new members to the assembly configuration while using the Place Member dialog.

The Autodesk Inventor Bill of Materials is enhanced to support iAssemblies displaying a single structure view that contains the aggregated item list. Each configuration is displayed as a separate column showing quantity (QTY) differences across members. Using the BOM editor you can display one member at a time or the entire set of members at once.

The Vault is enhanced to support assembly configurations allowing you to select and check out specific members of a configuration.
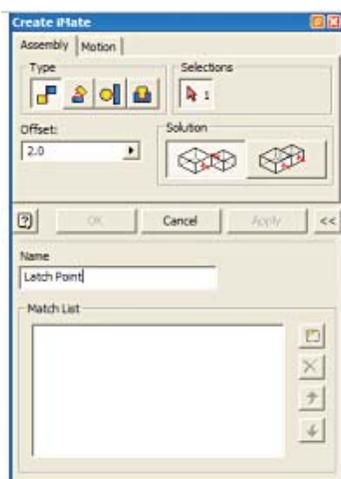
The Vault can locate the factory, or parent, file from a member and can also locate other files that make use of the member file using the where-used capability.

Note: The iPart table has been enhanced for the assembly environment by the addition of columns for items such as adaptivity, representations, constraints and assembly features. In addition, every row in an iPart/iAssembly table will have a unique name.

iMate Enhancements

Autodesk Inventor iMates have been updated to expand their capabilities and encourage increased usage. Enhancements include:

Improved Component Placement using iMates—in the current version of Autodesk Inventor, components are placed automatically based on the first iMate match in the target assembly. It is now possible to switch into an enhanced iMate placement mode from the context menu. This mode provides new context menu options for cycling through the iMates on the component as well as those on the target assembly. Together with improved graphical feedback, these options give you precise control over the pairing to be used for the component placement.

iMate Naming—to simplify naming of iMates, a new Name field is provided in the iMate Create dialog. With this field you no longer need to use the iProperties dialog to define the names of new iMates.

A new Match List allows support for multiple name matching to improve chaining of components and polarity of iMates. The new controls are located in an expandable area of the dialog. The same enhancement has been added to the iMate properties dialog to support modification of iMate names.

Infer iMates from Features—Autodesk Inventor R11 infers circular iMate candidates from new features with circular symmetry. In addition you can infer iMates from existing features using the new Infer iMates tool.
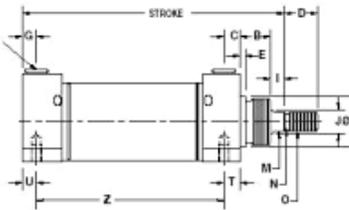
New Drawing Tables

A new Table tool is added to facilitate the documentation of assembly configurations by allowing you to create a drawing table that displays information from the parameter table of an iAssembly or iPart. The drawing table is linked associatively to the corresponding iAssembly or iPart so that its values update automatically whenever values are changed in the modeling environment.

Similar to the Parts List tool, the Table dialog allows you to select a view and choose which of the iPart or iAssembly columns to display. Table formatting is controlled by a new table style, and you can override the table style setting as with Parts List.

The table tool can also be used to create custom tables, such as revision tables, with information entered directly or imported from an Excel spreadsheet.

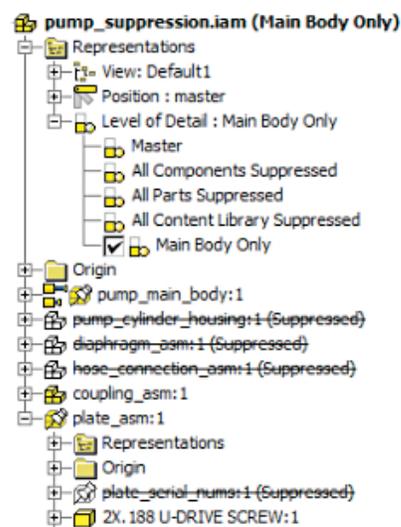| Bore	A°	B	C	D	
1-1/16" (09)	3.50	0.50	0.39	0.50	
1-1/2" (17)	3.87	0.63	0.46	0.75	
2" (31)	5.07	0.81	0.44	0.88	

Large Assembly Management

The Large Assembly Management tools allow a subset of components to be unloaded from memory enabling you to continue working on the remaining components. In this manner, users can realize much larger design goals, without being forced to load the entire design at one time.

A new Capacity Meter located at the right of the status bar provides continuous feedback on

resource usage including the number of occurrences in the active document, the number of files opened and the amount of memory in use.

Having suppressed one or more components you can create a Level of Detail (LOD) representation to save and reuse at a later time. You can create LOD representations to support different design activities. In addition all new assemblies have four default LOD representations: Master (which loads all components), All Components Suppressed, All Parts Suppressed and All Content Center Suppressed.

During File Open, Component Placement, and View creation, the Options panel allows selection of all three Representation types. By selecting an appropriate LOD representation you can quickly open very large assemblies and then open just the components you need to complete your work.

Shape Modeling

Part modeling has been enhanced to enable the design of cast and molded parts that meet design requirements for aesthetics, ergonomics, and manufacturing constraints. This is met through additional options for controlling the shape of lofted and swept features as well as new controls for edge continuity and tangency.

The Loft and Sweep features now provide you with high-fidelity previews that enable you to view the shape of the new geometry before committing to update the model.
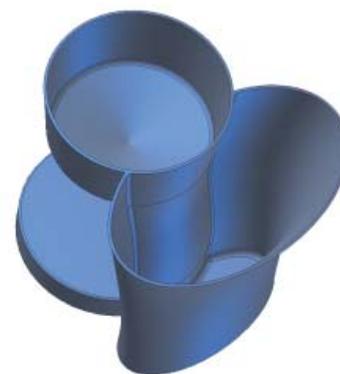
Advanced Loft and Sweep Options

Centerline Loft—the Loft feature is enhanced with support for lofts controlled by a single centerline rail. The centerline option creates shapes in which the intermediate cross sections are held normal to the centerline providing a more consistent transition between the user-defined sections. The centerline rail is not required to intersect the section curves.

G2 Loft—for closed-section lofts G2 conditions may be applied to the beginning and end sections (similar to tangency conditions). For open-section lofts G2 conditions may be applied to the beginning and end sections and to rails allowing for G2 continuity at each of its four outer boundaries.

Loft to a Point—the enhanced Loft feature provides the ability to loft to a sharp or tangent point, allowing loft sections to transition to a pointed or rounded tip. For tangent points, there is an option to override the inferred tangent plane with a user defined tangent plane.

Two Rail Sweep—the Sweep feature is enhanced to allow the selection of a Rail to control the scaling and/or rotation of the swept profile. As with a single rail sweep, the two rail sweep uses the Path, which must be G1 continuous, to control the normal of the Profile plane. The Rail acts as a guide curve, controlling both scaling and twist.
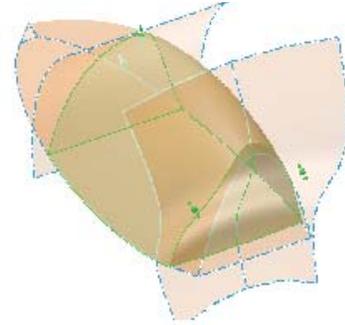
*Normal to Surface Sweep—*this is an enhancement to the single-rail sweep mode that maintains the profile normal to the path using a guide surface.

*Profile Orientation Control—*provides explicit control of the profile orientation; allowing the swept profile to be held normal to the Path, or to be held parallel to the original profile.

New Sculpt Tool

With the new Sculpt Tool you can quickly create solid part geometry by combining surfaces and faces. The only requirement is that the selected surfaces and/or faces must define an enclosed or "watertight" volume. You can create new standalone solids or you can add material to an existing part. With the Sculpt tool you can also use a surface to remove material from an existing part. Unlike the Stitch tool, the Sculpt tool does not require the surfaces to be trimmed to the common edges which results in much faster and efficient modeling.

Enhanced 3D Sketch Tools

The 3D Sketch tools have been enhanced to support the new shape modeling tools providing greater control over sketch geometry. The 2D and 3D panel bars now have consistent layouts and equivalent 2D and 3D sketch commands have been rationalized to provide a simpler user interface. The following new capabilities are provided:

*New Smooth Constraint—*establishes G2 continuity between a spline and another curve. The second curve can be a line, arc or another spline. Available in both 2D and 3D sketch environments.

*Project To Surface—*projects a curve onto selected faces. Projection options include along a vector, to closest point and wrap to surface.

*3D Intersection Curve—*creates a 3D curve from the intersection two planar curves. You can also create an intersection curve using one planar curve and a surface or two intersecting surfaces.

*Reorient the Precise Input Tool—*enables you to reorient the Precise Input Graphic by selecting a plane so that you can create a 3D sketch on the selected plane.

*New Spline Tangency Handles—*provides increased control of spline shape by manipulating tangency handles.

*Wrap Curve to Surface—*allows you to project curves onto cylindrical and conical surfaces while preserving curve length. This feature is useful when you need to create a precision length path for sweeps such as CAM type slots in faces.

Enhanced Surface Editing Tools

The following tools have been added for working with surfaces:

*Enhanced Boundary Patch—*the 3D boundary patch is used to fill an n-sided gap in the model. The gap may be in a solid or a surface model. It is useful for healing imported geometry or to create a water tight model for transfer to an FEA package.

*Replace Face using a WorkPlane—*the Replace Face tool has been enhanced to allow the selection of a work plane as a replacement face.
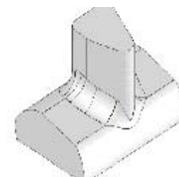
*Surface Pattern Mirror—*You can now mirror and pattern surfaces.

*Surface Trim/Extend—*There are new tools for trimming and extending surfaces

Enhanced Fillets

The Fillet tool provides two new fillet types: Full Round fillets and Face-Face fillets. The fillet tool now includes a high fidelity preview enabling you to visualize the resulting fillet.

Face—Face Fillet— option to create fillets between two sets of faces allowing the creation of fillet geometry which can not be created using traditional edge-based filleting. It also supports filleting between non-adjacent faces (faces which don't share an edge) and allows blending over small edges and other irregular geometry. Fillets between faces are generally more robust than edge fillets because faces are more persistent than edges during design changes.

*Full Round Fillet—*new option for rounding or "capping" ribs and other exterior part features. By connecting three adjacent face sets with a variable radius fillet, the Full Round fillet allows the creation of fillet geometry which is difficult or impossible to create using traditional edge-based filleting.

Edge Continuity—provides the ability to create curvature continuous (G2) Edge fillets that maintain curvature continuity to adjacent faces. Such fillets provide a smooth, continuous transition in curvature for use in generating aesthetically pleasing parts.

Rib With Draft—This feature will allow users to apply draft while creating the rib feature eliminating the need to apply the draft as an additional feature. The draft applies to all lateral faces created by the rib.

Approximate Shell Offset—addresses offset failures that occur when the requested offset is greater than the maximum curvature of the surface such as a loft to a sharp point. A solution for such cases is generated by calculating an approximate offset of the surface.

Usability

Browser-Extend Consumption—improves the user part modeling experience by reducing browser clutter, especially in surface modeling workflows, and for parts containing a large number of work features. In addition it helps you to identify dependencies in parts by indicating which surface and work features are used by other features

Assembly Constraint Diagnostics—the Design Doctor has improved diagnostics for analyzing constraint failures. On the Design Doctor dialog box, you select a new treatment named "Related Failed Constraints." This treatment enables you to isolate a smaller set of components that may participate in a constraint failure, thus helping you to find and repair errors.
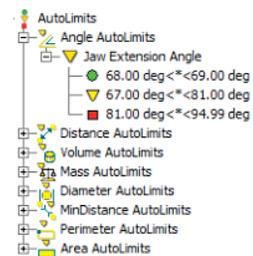
Comprehensive Analysis Tools

With the new analysis tools, you can check design data for surface quality and ease of manufacture. You can save multiple types of analysis with the model. Surface analysis and cross section tools are accessible from both the part design and construction environments.

- Zebra analysis, with density control and enhanced display accuracy, provides visual confirmation of surface continuity and tangency.

- Gaussian surface analysis provides feedback of surface curvature.

- Cross section analysis displays wall thickness with color-coded feedback of minimum and maximum thickness violations.

- Draft angle analysis displays color-coded draft angle based on a pull direction which can be defined by an axis, plane or planar face.

- Checks minimum distance between two components or faces in an assembly.
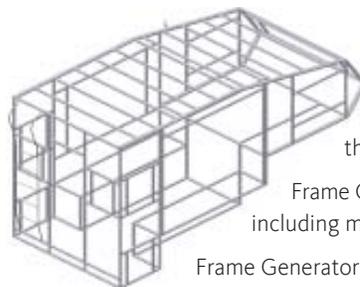
AutoLimits

AutoLimits are sensors for monitoring critical design parameters including length, distance, angle, diameter, loop length, area, volume and mass. You can create sensors at the document, feature, and sketch level indicating the parameter and the allowed limits. Once created, the AutoLimit sensor tracks the selected parameter and reports any changes that drive the parameter outside the user defined limits. AutoLimits provide visual warnings through a series of color coded graphic icons.

New Frame Generator

The new Frame Generator simplifies the design of structural machine frames as well as external frames for platforms, access ways and stairwells.

With Frame Generator's streamlined workflows you can quickly select, place and trim structural frame members. You can use both 3D wireframes and the edges and vertices of part models to define the location of structural members.

The Insert Profile and Change Profile dialogs provide enhanced multi-select and dynamic preview to simplify the placement and orientation of the members.

Frame Generator includes tools for commonly used end conditions including miters, notched profiles and trim to face conditions.

Frame Generator is integrated with the Autodesk Inventor Bill of Materials with support for lengths and consolidated counts of individual structural frame components in bills of materials, parts lists and balloons.

Drawing Manager

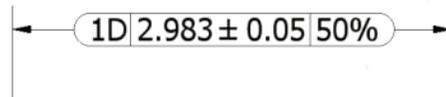A large number of enhancements are incorporated in the Autodesk Inventor Drawing Manger. Many of these improvements address requests and suggestions from customers. Enhancements have been made in the following areas:

- Dimensioning
- Drawing Standards
- Plotting
- Automation and customization through the API

Dimension Enhancements

Move dimensions —you can now move reference or model dimensions from one drawing view to another drawing view. Any dimension, including linear, diameter, radius, and angle dimensions, can be moved as long as attachment points are available in the target view.

Inspection Dimension —You can now convert a placed dimension into a DIN-standard inspection dimension and edit the shape attribute, label text, and inspection rate.

Diameter Dim Text Wrap Under Landing —when dimension text wraps to multiple lines, the first line is placed above the landing and the rest of the text lines below the landing line. This enhancement provides conformance to the GB Standard.

Retain Invalid Dimensions Option —annotations may become invalid if the component they are attached to is deleted, promoted, demoted, or replaced. New settings on the Document Settings dialog box, Drawing tab, let you specify your preferences for handling invalid annotations. You can highlight the feature-based annotations in a special color and preserve orphaned annotations.

Flip Dimension Arrowhead Direction —on drawing views, you can change the direction of one or both arrow heads to inside or outside the extension line.

Chamfer Note —there is a new Chamfer Note for adding chamfer notes to a drawing view. Chamfer notes may be attached to view and sketch chamfers. A chamfer edge and reference edge can be from different bodies, models, or sketches, but must be part of the same view.

Dimension to Midpoint and Intersection —on drawing views, you can now select midpoints of geometry or intersections of overlapping model edges when creating or reattaching dimensions. A midpoint is indicated by a green dot and an intersection is indicated by a gray dot. This enhancement eliminates the need to create construction geometry to define the midpoint as well as the extra commands to define an intersection.

Improved Edit Dimension Workflow —The dialog boxes for dimension text and tolerances have been combined into a single Edit Dimension dialog for editing dimension text, precision, tolerances and inspection dimension format.

Copy Dimension Properties —similiar to the AutoCAD Match Properties command, the new Copy Properties tool provides a quick and easy way to select a source dimension and apply its properties to other dimensions.

Hole Note in Side Views —hole note creation, manipulation and update is enhanced to support Hole Notes from linear edges in side views of holes in base, projected, detail and section views. The Hole Note will attach to the edge selected by the user. If an edge is no longer available after a view has been updated, the Hole Note will reattach to the outermost edge of the originally selected feature.

Improved Workflow for Ordinate Dimensions —create ordinate and baseline dimensions with fewer mouse clicks by selecting the target geometry with a window-select. This enhancement applies to the Ordinate and Baseline Dimension tools (including the Ordinate Set and Baseline Set tools).

Annotation Support for Feature Suppression —dimensions and other annotations are now suppressed if attached to features that are suppressed or parts that made invisible. Annotation will consistently become suppressed and unsuppressed as model geometry is suppressed and unsuppressed.

Standards Enhancements

These enhancements reduce the work required to create drawings that comply with drawing standards eliminating a number of cumbersome workflows and reducing the time required to complete production drawings. The enhancements include:

All-around Symbol Size—all-around symbols are now controlled from a single default setting in the Leader style. The symbol is used for feature control frames, surface symbols, and weld symbols.

Standard Style Scale List—the Default Standard dialog in the Styles and Standards Editor contains new preset values for drawing scale using decimal, fractional and rational formats. The preset scale values are used in the view creation and edit dialogs. Scales are now stored as strings instead of real values to maintain accuracy with non-rational numbers. When placing or editing views, you can also override the preset scales by typing directly into the Scale control.

Configurable Front View Plane—there is a new Front View Plane option in the Default Standard which provides control over the plane used to define the orientation of the Base View. You can select the YZ, XZ or XY planes.

Object Defaults in Standard Style—the Object Defaults belonging to the Standard Style are now displayed on a third tab in the Standard Style dialog. An edit button provides a shortcut for editing the selected Object defaults style.

Enhanced Drawing Templates—revision tables, placed user defined symbols and general note text are added to the list of items that can be saved in a drawing template.

Enhanced Character Picke—new options to insert font-based diameter, plus/minus and degrees symbol and a new link for opening the Windows character map

Multisheet Plot

The new Multi-Sheet Plot tool simplifies the process of composing and printing multiple sheet plots. The tool provides a simple to use wizard that guides you through the process of selecting and setting up the printer and selecting the files to be plotted. It generates a composite, or layout, with a preview of each plot sheet. You can submit the plot to a selected printer of save the composite as a batch job for the Task Scheduler. You can also schedule a previously composed multi-sheet plot via Task Scheduler.

Drawing Manager API

The Drawing Manager API is extended to support the development of a wider range of custom applications.

Drawing geometry is now exposed in the API with new query functions to access and highlight edges in the drawing views. Using these functions custom applications are able to navigate the drawing data model and correctly associate new annotations to the existing drawing objects.

Autodesk Inventor 11 provides full API support for creating, modifying and selecting circular and rectangular detail views, general dimensions, leader text and sketched symbols with or without leaders. The API includes support for attaching new detail views and leader text to existing geometry.

Enhancements to enable improved user interaction while in a custom command include new drawing object filters in the InteractionEvents object to enable the client application to enforce picking of only appropriate object types.

Bill of Materials

Bill of Materials management has been enhanced in three key areas: BOM editor ease of use, persistent item numbers, and support assembly configurations.
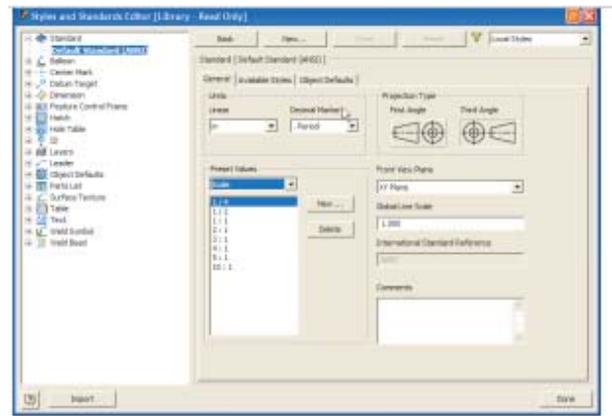
With the inclusion of BOM item numbers we now support associative item numbers for parts lists.

Bill of Materials Editor

Enhancements to the Bill of Materials editor provide improved usability and simpler workflows for modifying item numbers, material properties and custom iProperties.

Material Editing—material properties, including weldment material properties, can now be modified directly within the BOM editor. Selecting a material cell displays a dropdown list containing the same material properties that are displayed in the Physical tab on the part's Properties dialog.

The material properties cells support multiple-selection allowing you to quickly modify multiple components in a single step.

Display Model Structure—access to the Model Data view has been moved to a new tab on the BOM editor to distinguish it from the BOM Views.

Virtual Component Material Properties—virtual components now support material properties allowing mass to be calculated from volume and material data. You can assign material styles to virtual components and you can override the calculated mass with a different value in the same way you can for other components.

Add/Create Custom iProperties—missing component iProperties are automatically generated when values are entered.

Optional BOM Views—new options allow you to disable the BOM views (both structured and parts-only views) when not required thereby reducing the overhead associated with creating and updating the BOM structure. Templates for assembly and weldment support the new BOM view options.

BOM View Properties—new properties are introduced for BOM Views. View properties enable you to control numbering rules for new items and are also used by the Renumber Item tool.

Edit Merged Item Quantities—you can now override a merged item's quantity using the new Show button to temporarily dissolve merged items. Make changes to the quantities of the merged items and then select the Clear button to exit from the Edit Merged Item state.

Item Number Synchronization

The item number is now associatively maintained between the Global BOM and the Parts list. The item number can now be edited as needed in both the Global BOM and in the Parts List. Changes to item numbers in the Global BOM are automatically updated in the Parts List and Balloons, except for values that have been specified as Static in Parts Lists. Static overrides to item numbers in the Parts list can be pushed back to the Global BOM to maintain associativity.

Enhancements have been made to the renumber command. A new Renumber dialog box offers selection controls to filter the items to be renumbered. A start value and increment provide additional control of number assignment.

The new Part Number Row Merge settings command allows you to control how components with the same part number are treated. You can choose to disable merging of items with the same part number and you create a list of specific part number values to exclude from merging.

The ability to modify an item number in the drawing and push changes back to the BOM makes it possible to retain local changes or share them across other documents. The browser indicates if the Parts List contains overridden Detail IDs.
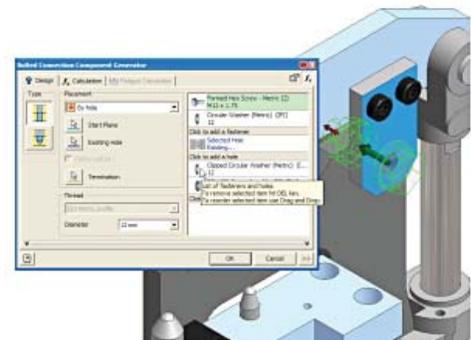
Design Accelerators

All of the Component Generators and Mechanical Calculators are easier to use with new interfaces that conform to the standard Autodesk Inventor look and feel. Information is organized by tabs that separate geometry selection from the design calculators.
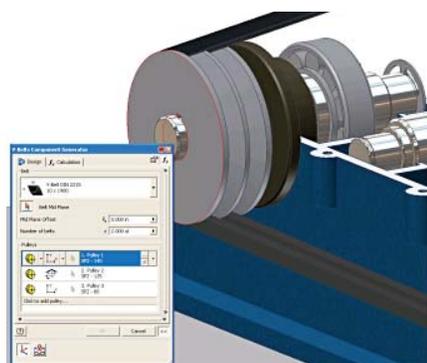
Bolted Connections

With the redesigned Bolted Connection Generator the design of and placement of bolts, washers and nuts is simple, efficient and flexible.

The Design tab of the Bolted Connection is integrated with the Content Center simplifying the selection of each component. The Design Accelerator automatically selects the correct component size and displays a preview of the complete connection. You can override the recommended component at any time by selecting a different component from the Content Center. The preview includes 3D grips which you can use to adjust the length of the bolt.

Power Transmissions

The Component Generators included in Autodesk Inventor 11 provide a comprehensive set of tools for the design of power transmissions for industrial equipment.

Quickly design drive shafts and then add bearings, circlips, keys and keyways, V-belt and pulley drives and gears. Adding these components is faster with the improved dialogs, dynamic sizing and graphic preview which snaps to the assembly.

The V-Belt generator streamlines the design of belt drives with options for fixed, sliding and virtual pulleys, single or multiple belts, and easy adjustment of the belt's mid-plane.

The new Component Generator dialogs enable standard graphics window operations, such as zoom and rotate, to facilitate identification and selection of geometry. In addition you can adjust the size of the dialog bars to reveal more of the underlying graphics window.

A new Settings option of each of the Design Accelerators allows you to define your own display names and file locations for the generated components.

The Engineer's Handbook is now directly accessible from the Design Accelerator menu panel.

Content Center

Improved Content Browsing and Selection

The Content Center browser is redesigned to simplify component searches. Filters are provided for each of the supported component standards and you can create your own filters to restrict the search to specific component types within each standard.

An optional Family Preview shows the members of the selected family in table form displaying the parameters of every available size.

There is a new Favorites folder in which you can place shortcuts to frequently used components and component folders.

The browser is fully resizable and the Tree View, Favorites View and Family Preview views can be turned on or off from buttons on the tool bar.

Content Center Publishing Tools

New authoring tools are provided for publishing content for use with Component Generators and Content Center. The authoring process guides you through creating and mapping appropriate parameters and iMates for each of the content based Component Generators and the Autodrop functionality.

There is a new Batch Publish tool which drastically reduces the time required to load multiple items into the Content Center.

There are enhancements to the tools for replacing family part templates and family edit enhancements

Content Center Library Management

Content is now stored in separate libraries for each component standard giving you the ability install only those libraries that you need.
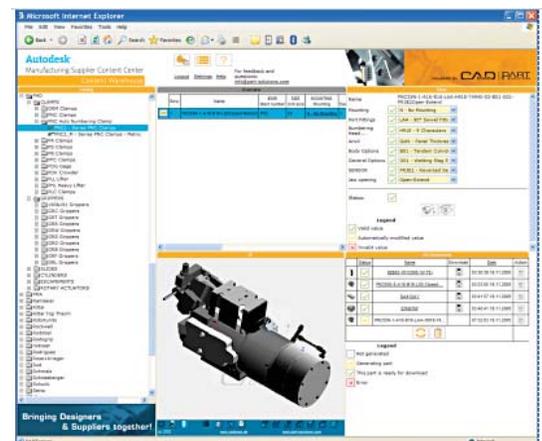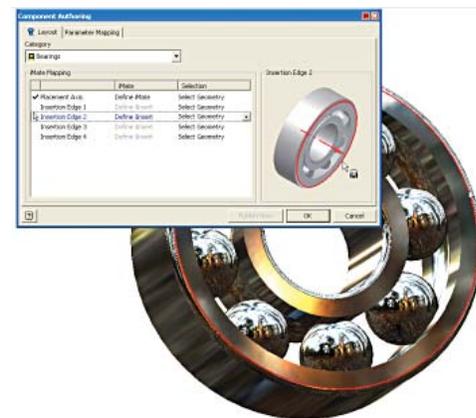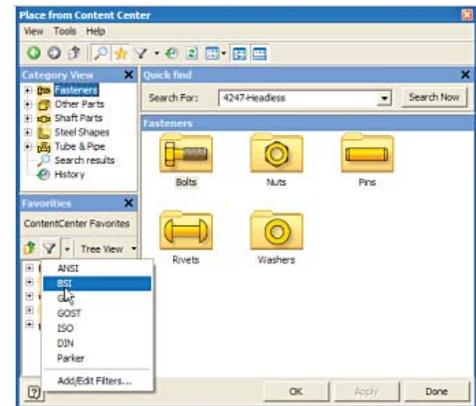
The content libraries are protected from inadvertent modification with full access control through database roles and permissions. You can now use the Autodesk Data Management Server tools to manage the content libraries providing improved administration and security.

Supplier Content Center

Be more productive with the new Autodesk Manufacturing Supplier Content Center. Incorporate purchased parts in your design without spending time modeling the parts yourself.

The Supplier Content Center provides you with access to one central location for manufacturer-specific content from leading suppliers worldwide.
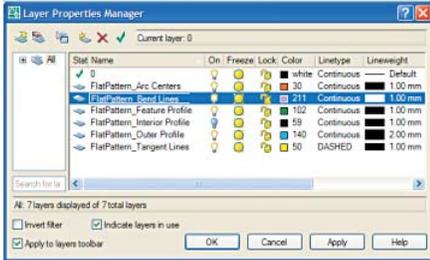
- Design products faster with easy online access to millions of native Autodesk Inventor models.

- Reduce errors by bringing in accurate, up-to-date component models.

- Build your own internal Autodesk Inventor Content Center by adding commonly used parts from your favorite suppliers.
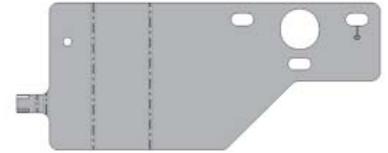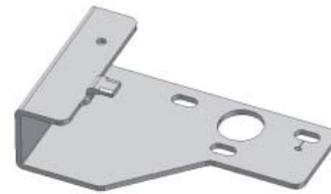
Sheet Metal

Autodesk Inventor 11 includes a new sheet metal unfolder which uses the Autodesk Shape Manager to calculate bend deformations.

The new implementation provides better performance and also accurately unfolds parts containing consecutive bends, interior tabs, non-parallel face cuts and features that span bends.

Output of flat patterns in DXF format is supported with separate layers for outer, interior and feature profiles, as well as bend lines, arc centers and tangent lines.

You can control DXF output, including the use of spline approximation and associated chord tolerance, by editing the XML template that controls DXF post processing.

Autodesk Inventor Studio

The focus of Autodesk Inventor 11 is enhanced usability with numerous improvements to Studio workflows that reduce the number of steps required to setup and generate renderings and animations.

Workflow improvements include:

- Save time using the rendering preview to see how the animation will look before running the full rendering
- Improved animated fade preview
- Improved constraint animation playback
- Animation between positional representations
- Support for the new LOD representations.
- Multiple animations per document
- Turntable animation
- Output by frames
- Copy / paste / multi-select for actions
- Mirror actions
- Save while inside Studio

Rendering improvements include:

- Rendering of decals and cosmetic threads.
- Better performance and less memory consumption during rendering.

Task Scheduler

There is a new Task Scheduler icon in the Windows "taskbar" which provides quick access to the most frequently used Task Scheduler functions.

Autodesk Inventor will now display a progress bar when a task is executing so that you can monitor its status.

The Task Scheduler provides support for several new tasks including batch printing, vault download and vault upload.
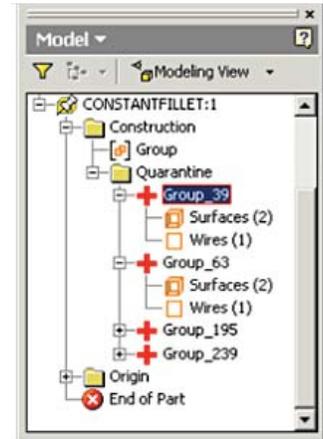
Transportation Supply Chain

Autodesk Inventor 11 delivers an integrated set of tools to support the design of manufacturing tool using imported design files. These tools enable you to preserve the original design intent when working with imperfect datasets. Imported geometry can then be enhanced with additional design intent required to develop accurate tool and fixture models.

Improved Performance for STEP and IGES Import

The capacity and performance of the STEP and IGES import tools has been improved enabling you to work with larger datasets even if they contain corrupted geometry. When importing assemblies, the import tools now save and unload the part files as they are translated. This reduces memory consumption and allows you to import larger datasets.
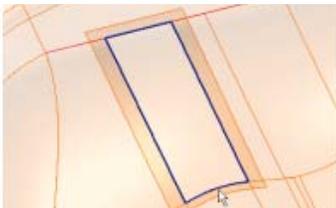
With Autodesk Inventor 11 all data is imported regardless of the quality encountered with defective data grouped in the new quarantine folder to facilitate subsequent repair. A new Translation Report shows each file name, the status of the translation and any error codes generated as a result of translation problems. In addition, the import function now processes multiple surfaces as a single feature and preserves color information in the imported data.

Surfaces can now be imported directly into the part modeling environment as a single composite surface feature making it easier to change properties and reduce browser clutter. Surface color information is maintained during translation.

Enhanced Construction Environment

The Construction Environment is enhanced to simplify the analysis and repair of problems encountered in the imported data and to enable workflows using surfaces and wire geometry.
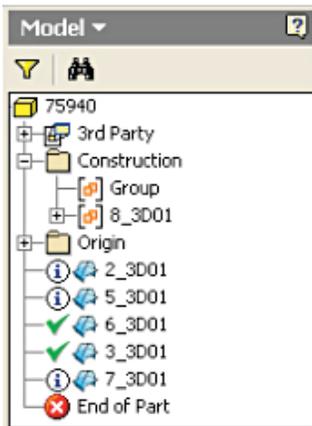
Surface Repair Tools—new Extract Boundary Loop and Boundary Trim tools make it easier to repair defective surfaces by using adjacent surface edges.

Stitch Tolerance—control over stitch tolerance, or gap, is available in the Stitch tool to provide more control over the stitching with less errors.

Refit Face—you can now recover bad surfaces using by refitting to a user specified tolerance.

Measure and Analysis Tools—are now available in the Construction Environment.

Promote Multiple Surfaces—a new Composite Feature saves time when working with multiple-surface datasets allowing the surfaces to be transferred to the modeling environment as a single unstitched body. The composite surface feature provides better performance than individually promoted surfaces.

Promote Wire Geometry—a new Promote Wires command allows imported wire data to be transferred to the both the part and assembly modeling environment and used as "lightweight" reference data. Wire data is also available in drawing views and you can easily change the appearance of the wire data in a drawing with the Select as Edge command now enabled for 3D sketches.

Color for Solids and Surfaces—this enhancement provides the ability to control color overrides for CE solids, surfaces and part modeling surfaces.

Enhanced Work Features

The new work features in Autodesk Inventor 11 facilitate the placement of tools and fixtures by defining points and axes relative to imported model geometry.

Create Axis Normal to Surface

you can now create a work axis normal to surfaces, including non-planar surfaces, by using a work-point that does not need to lie on the surface.

Work Point on Loop Center

places a work point at center of imported data "hole".
This feature enables the ability to create a work point in the center of a non-analytic hole.

3D Sketch Points

you can now add sketch points to a 3D sketch using the 3D sketch triad. You can also create 3D sketch points based on the vertices of 3D geometry or by constraining to a drawing plane using the precise input toolbar.

Work Axes Through Work Points

allows a work axis to be placed through a Work Point and keeps it parallel to any selected sketch geometry or edge. Use this to define the location pins for tooling design.

Work Axis Length

you can now adjust the length of the work axis in a part or assembly.

Import and Use Point Data

Autodesk Inventor 11 makes it much easier to work with point data which is often used to define the location of spot welds. New tools are provided to import 2D and 3D point data into the sketch environment. Point data can be easily projected onto a surface or along a surface normal.

Break Link in 3D Sketch

Break Link will is added to all 3D sketch reference geometry. It works in a similar manner to 2D sketch Break Link. After the link is broken, any reference geometry remains in the 3D sketch as un-associative sketch geometry.

DWF Output

DWF output is enhanced to enable the 3D model to be output to the same file as the drawing sheet. New options allow you to select the drawings sheets and to control whether or not to include the 3D data.

AEC Exchange

Use AEC Exchange to quickly prepare and publish Autodesk Inventor assemblies as content compatible with 3D building design software used by the professionals in the Architectural, Engineering and Construction (AEC) industry. With AEC Exchange you can define intelligent connection points and output the appropriate native file formats for consumption by Autodesk Building Systems. What's more, you can also output 3D DWG geometry to Autodesk Architectural Desktop, Revit, and AutoCAD.

Autodesk Inventor Professional Functionality

New Autodesk Inventor Professional Editions

With Autodesk Inventor 11, Autodesk is pleased to announce the availability of two new product configurations of Autodesk Inventor Professional. Each edition offers Autodesk Inventor Professional capabilities to meet the needs of specific engineering functions:

The two new editions are:

AUTODESK INVENTOR® PROFESSIONAL FOR SIMULATION
Includes the new Dynamic Simulation functionality and Finite Element Stress Analysis

AUTODESK INVENTOR® PROFESSIONAL FOR ROUTED SYSTEMS
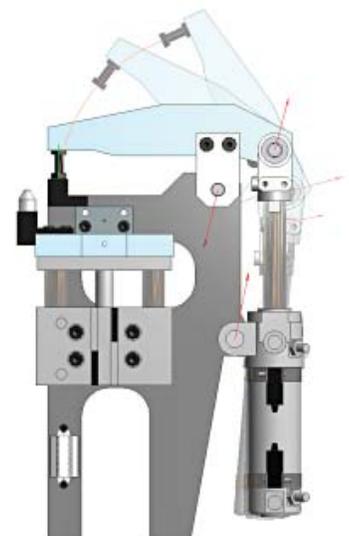Includes Tube and Pipe Design, Cable and Harness Design and IDF import.

The existing AUTODESK INVENTOR® PROFESSIONAL will continue as before providing all of the Inventor Professional tools in one package.

Dynamic Simulation

New in Autodesk Inventor Professional 11 is Dynamic Simulation. With full motion simulation you can now simulate the operation of mechanisms and motorized assemblies to make sure your designs will work while reducing physical prototyping costs.

Motion Simulation

Compute the dynamic operating conditions of your design throughout its full operating cycle and accurately size motors and actuators to sustain actual operating loads. Provides a complete analysis of the positions, velocities, accelerations and loads encountered by each component of the mechanism.

Output to FEA
predict stress and deflection with accurate peak loads by transferring reaction forces from discrete time steps to the Autodesk Inventor Stress Analysis or Ansys Workbench.

Motion Constraints
identify relations between moving parts from a palette of dynamic constraints then add springs and dampers and define the coefficient of friction that applies at each joint.

Load Definition
investigate your design's performance under a range of different load conditions by applying different driving loads and moments as well as time-based force functions using the load profile editor.

Visualization
understand the behavior and performance of your design using the animated 3D visualization to display dynamic motion based on underlying physics models and the applied load conditions.
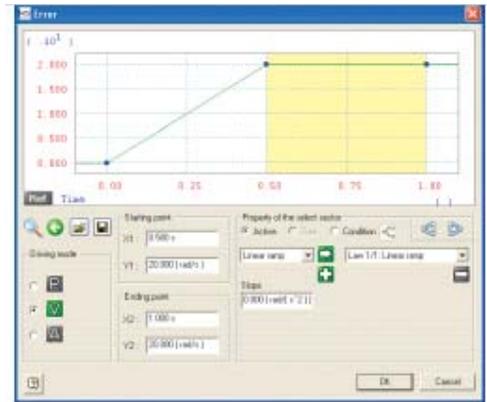
Point Trace
validate component position through the operating cycle by using the Trace option to display the path of selected points.
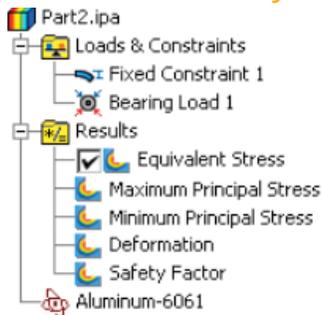
Graphing
use the comprehensive graphing capability investigate how properties such as force and acceleration vary through the machine's operating cycle.

Excel Output
export XY plot data to Excel worksheets for use in presentations and reports.

Finite Element Analysis

Finite Element Analysis has a new option to display maximum and minimum principal stresses.

Autodesk Inventor Professional 11 introduces a new dialog to control the legend (color bar). This dialog replaces the in-window color controls used in previous versions.
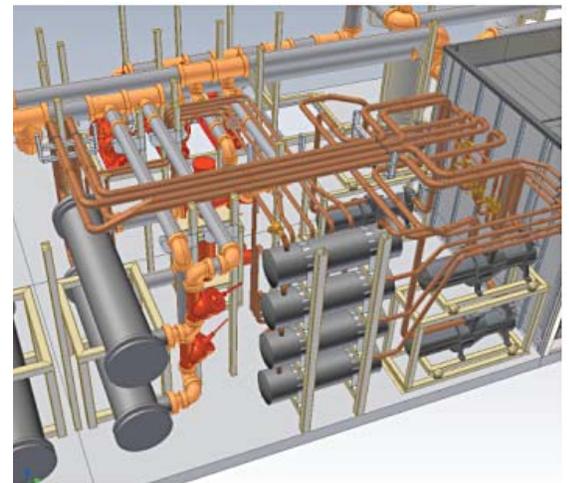
The stress analysis tools are compatible with the new Dynamic Simulation tools allowing reactions forces to be transferred into the stress analysis environment

Tube and Pipe

Parametric Routes
The Tube and Pipe environment within Autodesk Inventor Professional has been upgraded to support a full parametric design paradigm utilizing 3D sketch geometry. With new auto-route regions, you can specify which sections of the pipe route are to be controlled by the auto-route engine and which sections are to be controlled by sketch constraints and dimensions.

The use of the 3D sketch tools gives much greater control over the pipe route enabling you to constrain it to other geometry in the design. This is very useful for maintaining pipe runs parallel to parts of the assembly and for establishing parametric relationships such as clearances and fall angles. Once established, parametric constraints increase the stability of the pipe routes when changes are made to the assembly.

Component Copy and Reuse
Copy Route and Copy Run, in conjunction with the new Make Adaptive command gives you the ability to quickly and easily reuse a complete pipe routing from a previous design. With Autodesk Inventor 11 the application now uses adaptivity to determine which occurrences are primary, driving occurrences and which are secondary, driven occurrences. Only occurrences that are marked as Adaptive can be edited with the routed systems tools.

Branch Fittings

Autodesk Inventor Professional 11 supports the insertion of attached fittings such as Weld-O-let fittings. These components are stored in the Content Center with the other tube and pipe fittings. The insertion process provides you with 3D controls for defining the linear position and orientation of the fitting. Once the location is defined, Autodesk Inventor creates the correct sized attachment hole in the pipe based on information contained in the fitting itself.

Make Adaptive Command

The Make Adaptive command converts a non-adaptive occurrence into an adaptive occurrence so that it can be edited with the normal design workflows for routed systems. It is available on non-adaptive occurrences of Master Runs, Runs, Routes, Hose Assemblies and Harness Assemblies. Similar to the existing Copy Components command, the Make Adaptive command replaces the secondary occurrence with a new assembly by means of a file-level copy of the structure underneath the target occurrence.
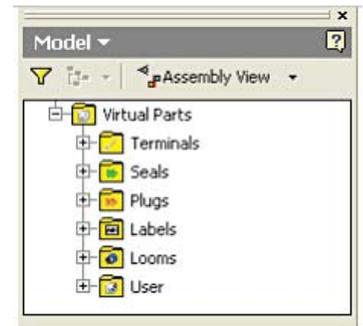
Cable and Harness

Virtual Parts

Define all of the components in the wiring harness, including terminals, seals, labels looms and plugs, by adding virtual parts to harness objects. Looms are rendered in the 3D model and labels are rendered in both the 3D model and the nail board view.

The Autodesk Inventor Bill of Materials recognizes all harness objects including wires, cables and virtual parts in order to provide a complete BOM for the harness.

You can also add user-defined virtual parts to the Cable and Harness library.

Wirelist Import-Export Tool

The wirelist import tools have been enhanced to facilitate the importing of large datasets with incomplete reference information. The import tool now detects missing connectors, pins and wire definitions and displays the wirelist enabling you to see which reference designators and pins do not have a counter part in the harness assembly as well as wires and cables that do not have a definition in the library. You can then edit the list to add missing information before completing the import procedure. The import tools can be used with wirelist data from AutoCAD Electrical and with data from other electrical design applications.

You can now export wirelist data in either XML or CSV formats for schematic design in AutoCAD Electrical.

Other Functionality

Windows XP Professional for x64 Edition

You can now run Autodesk Inventor on personal computers that use the x64 processor architecture: AMD64 or Intel EM64T. Autodesk Inventor 11 is fully certified for operation on Windows XP Professional x64 Edition. It will run as a 32-bit application using the WOW architecture of Windows x64. This increases the available virtual memory to the full 4 Gigabyte limit for a 32 bit process.

Direct3D

Autodesk Inventor 11 supports Microsoft Direct3D graphics drivers. The Autodesk Inventor installer sets Direct3D as the default driver during installation. If you prefer to use OpenGL you can switch drivers on the Hardware tab of the Tools —Applications Options dialog.

Direct3D is the framework for current and future graphics development on Microsoft Windows. As a widely used Microsoft standard it enjoys better support from the graphics hardware manufacturers and is less prone to driver support issues than OpenGL. For this reason you should expect substantially fewer driver compatibility problems when using Direct3D.

Licensing

New cascading network licensing supports customers who need to deploy multiple Autodesk products. For example, AutoCAD 2007 licenses can easily be combined with licenses for Autodesk Inventor. As AutoCAD Mechanical licenses are used, the network first chooses the stand-alone AutoCAD Mechanical licenses, then the Autodesk Mechanical Desktop licenses, and finally the applicable Autodesk Inventor licenses. The manufacturing offerings from Autodesk fit together to form a complete solution.

Installation

The new, updated installation interface helps users to install a group of products more easily than ever before. The older custom install dialog boxes have been replaced with an interface that makes it easier to make decisions about your installations choices. For example, the dialog box to the right shows all of the different standard part libraries that you can choose to install. The three main components of the installation can all be chosen at one time, including AutoCAD Mechanical 2007, Autodesk Vault 5, and the Autodesk® DWF™ Viewer. The Vault Server can be installed on a local workstation or in a central location that all users can share. New documentation guides users through the process and helps new data management users find the Vault server to begin managing work-in-progress data.

Stay Up to Date

Autodesk gives you more. Gain access to technical expertise, utilize training and support programs direct from Autodesk, learn about the latest product releases and give us your feedback about the Autodesk Inventor. Not only does Autodesk want to help you use Autodesk Inventor more effectively but we want to make sure Autodesk Inventor is working effectively for you.

Subscription

Log into the Subscription Center to receive product updates, log and view support requests, or take eLearning courses. A valuable component of Autodesk® Subscription, eLearning provides a continually expanding curriculum of short training exercises.

Product Updates

If you experience an issue with Autodesk Inventor 11 that has already been solved in a service pack or hotfix, a dialog box appears when you submit the problem to Autodesk, enabling you to immediately install the new service pack or hotfix.

Feedback

Autodesk Inventor 11 customers can provide feedback to the Autodesk Inventor 11 development team through several different avenues:

- Provide tips or join newsgroups at www.autodesk.com/inventor

- Keep up to date on what's happening in your industry, stay in touch with other industry professionals, and take advantage of a host of online resources at the Manufacturing Community Portal at mfgcommunity.autodesk.com

- Talk with your Autodesk Authorized Reseller and support staff

Your input is crucial to our success and we look forward to receiving your suggestions.

Conclusion

We thank you for your continued support of the Autodesk Inventor family of products and hope you feel we are listening to your needs. The new and enhanced functionality was done to make you more productive, your company more competitive, and return true value to your bottom line.

Autodesk®