Technical What's New

Autodesk® Inventor™

2009

Inventor Suite

Inventor Simulation Suite

Inventor Routed Systems Suite

Inventor Professional

Image courtesy of HTC Sweden AB

Autodesk

Technical What's New

Contents

Technical What's New

Autodesk® Inventor™ 2009 - Removing Barriers to Productivity

Welcome to Inventor 2009

Autodesk Inventor 2009 offers significant increases in productivity especially in the core design tools that you use every day. We've listened to many people as we put this release together and many of the improvements are a direct result of your input. Inventor 2009 is easier to use and more powerful than ever, providing a more accurate digital prototype so you can understand how your design will perform before you build it.

We hope this will help you to analyze the new features so you can focus on those that will have the biggest impact on your operations.

The Inventor Team

With over two hundred separate enhancement projects in the release, the Autodesk Inventor 2009 product line addresses three key themes: working on larger designs, enabling more innovation and working with other people's data. Within each theme we've outlined the key enhancements so you can focus on those that will have the biggest impact on your operations.

Working on Larger Designs

Digital prototypes of large scale designs such as industrial plant, specialty vehicles, or modern transportation systems demand the ability to construct and manage large assemblies which can run up to tens or even hundreds of thousands of parts. Inventor 2009 introduces native 64 bit support which removes the memory barriers associated with the Windows XP 32 bit platform. Together these two factors facilitate the design of architecture so working with large digital prototypes won't slow you down.

- Native 64 bit removes 3GB limitations and support PCs with more RAM
- Component substitution provides reduced memory representations of complex sub-assemblies
- Performance optimization – including start up, shut down, and many modeling operations

Enabling More Innovation

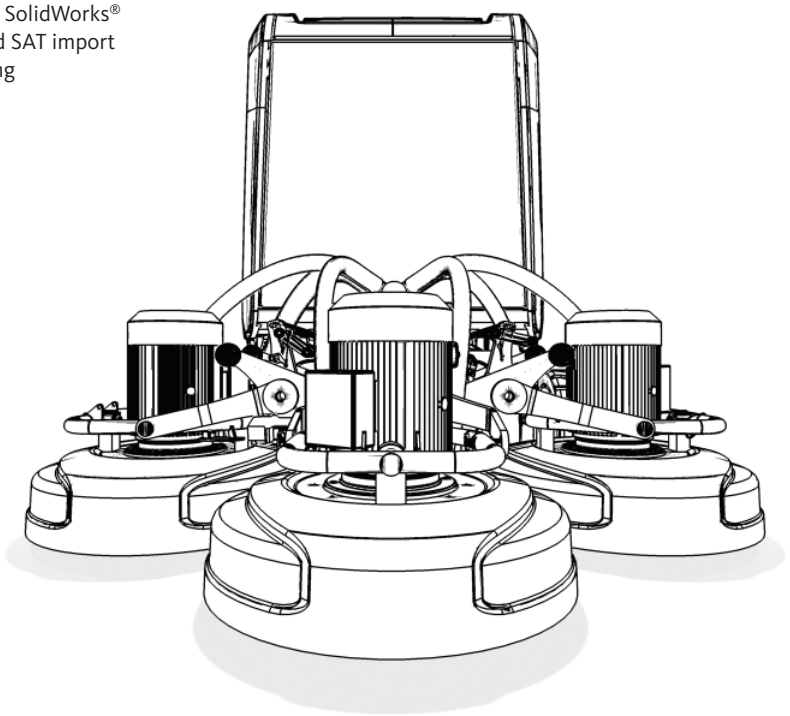
Improvements to the sketching, design accelerators, part design, and sheet metal simplify the design workflows while enhanced simulation tools give you the power to explore more design options and evaluate performance and efficiency without building physical prototypes.

- Easy to use simulation tools integrated with the design workflow
- Advanced sheet metal functionality that incorporate manufacturing factors into the 3D model
- Smart pipe runs with new support for hygienic pipe standards

Working with Other People's Data

In today's world of integrated supply chains, designers increasingly need to work with 3D data from customers and suppliers. Avoid the time-consuming difficulties of working with data provided through "neutral" formats by reading data directly from the original or native formats.

- Native translators for Pro/ENGINEER®, Granite, UG-NX™, Parasolid®, and SolidWorks®
- Enhanced STEP, IGES and SAT import
- Enhanced DWF Publishing



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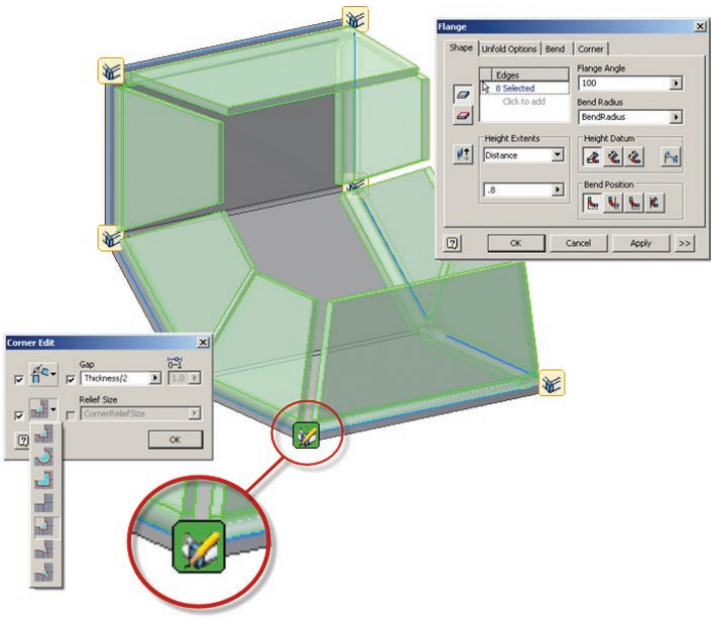
Sheet Metal Design

The Inventor 2009 product line is designed to simplify the design of a large range of widely used sheet metal constructs by providing accurate digital prototypes that faithfully represent the as-manufactured part including such details as corner relief geometry, sheet metal fasteners, and bend conditions calculated from both linear equations and empirical bend tables. Bend tables are now incorporated in revised sheet metal styles that are fully integrated with the Inventor Styles system. These changes make it easier to define and manage sheet metal styles to reflect actual material, gauge, and manufacturing variables used within the selected manufacturing facility. Building on the drawing automation introduced with Inventor 2008, Inventor 2009 introduces punch notes that accurately document the punch data in manufacturing drawings as well as new options that support the documentation of flat pattern extents.

Corner Relief & Corner Seam Override

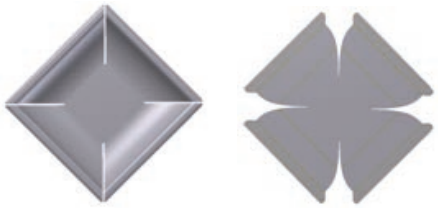
Quickly create complex flanged parts in very few steps and with a much smaller feature tree. The Flange and Contour Flange tools provide new direct manipulation capabilities so you can apply overrides to individual corner seams and corner reliefs in multi-edge flanges and contour flanges. Corner relief options support shape and size overrides and corner seam options support overlap type, gap value, and overlap percentage[1]. With fewer features and new direct manipulation glyphs (icons) that are revealed automatically in both create and edit modes, Inventor 2009 sheet metal parts are easier to build, easier to understand, and easier to modify.



[1]Overlap percentage applies to flanges only.

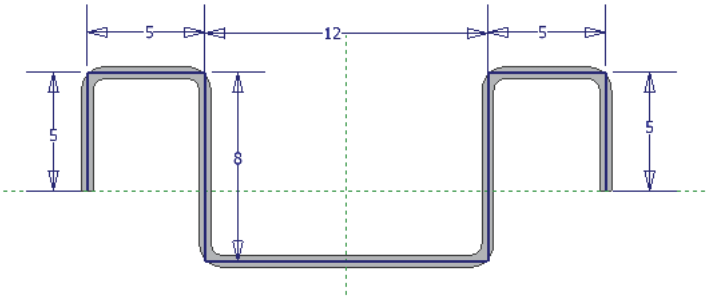
Large Radius Bends

Create accurate geometry in the presence of large radius bends. Flange, contour flange, and corner seam features generate miters with a consistent gap along the length of the seam for large radius bends. This is facilitated by a new corner relief shape called Arc Weld.



Contour Flange Mid-Plane Offset Option

Reduce the time required to create complex contour flanges. The Contour Flange has a new offset option that treats the contour profile as the mid-plane. When this option is selected, material is created on both sides of the contour profile.



Sheet Metal Styles

Set up and share sheet metal styles for different materials and sheet gauges. Import bend data from Bend Table Definition Files and manage with the Inventor Styles Editor. With Inventor 2009, sheet metal styles are fully integrated into the Inventor Styles framework. Sheet metal styles information is divided into Sheet Metal Rules and Sheet Metal Unfold Rules.

- The new Sheet Metal Rules define the bend and corner relief options that govern the design and behavior of sheet metal parts.
- The new Sheet Metal Unfold rules capture your commonly utilized unfolding methods and allow you to publish them to Inventor's style library. Sheet Metal Unfold rules can be created for both linear and bend table based unfolding methods. Enhancements in Inventor 2009 allow you to import and visualize legacy bend tables, edit them natively within the Style and Standard Editor and publish them as either .xml files for use within Inventor's style library, or as .txt files for use in legacy versions of Inventor.

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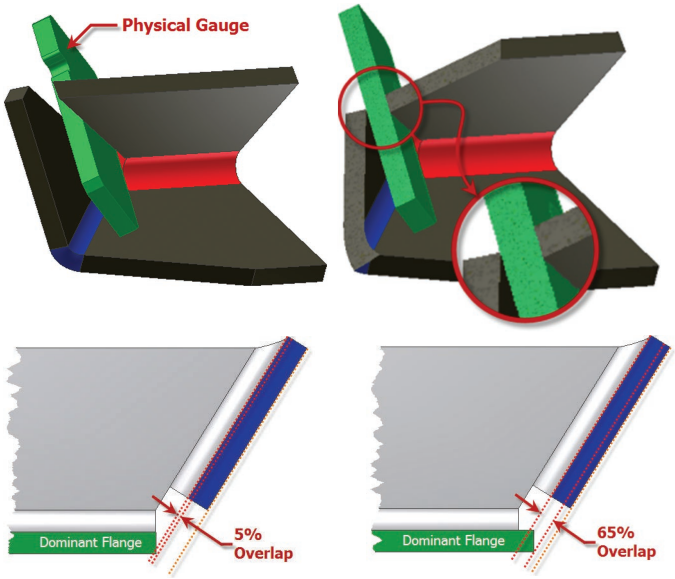
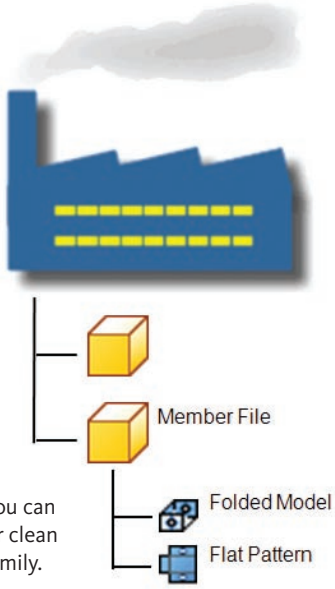
Enhanced Corner Seam
Impose precise control over corner seem gaps to insure correlation between gap value entered and manufacturing inspection processes. The new maximum gap condition (Max. Gap Distance) maintains a constant gap along the seam to improve the quality of seam welds. Use the new Overlap options to control percentage overlap between the dominant and submissive flanges.

As with the enhanced Flange and Corner Flange tools, the Corner Seam tool supports large radius bend plate cases, arc weld corner reliefs, and overlap adjustment.



Sheet Metal Fasteners
Insert PEM® brand fasteners, including self-clinching nuts, pressed nuts, standoffs, and studs into sheet metal designs. The Content Center has a new category of sheet metal content containing a large selection of frequently used PEM brand sheet metal fasteners. The new content can be utilized in sheet metal assemblies through enhancements to the Bolted Connection and AutoDrop tools.

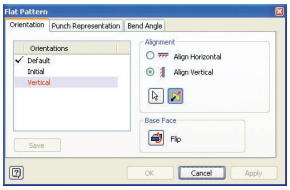
iPart Authoring for Sheet Metal Parts
Create families of related sheet metal parts. In addition to variations found in standard iParts, Sheet Metal iParts have the ability to reference Sheet Metal Rules and Sheet Metal Unfold Rules. By referencing these into iPart members, you can easily create a family of parts that represent the geometry to support the manufacturing capabilities of different suppliers.



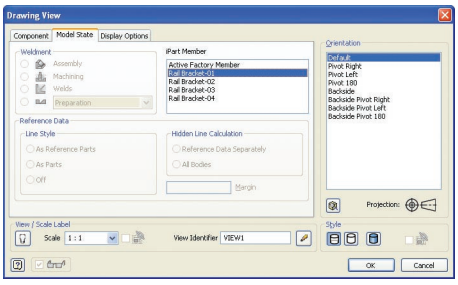
Sheet Metal iParts now include both the folded and flat pattern models so that you can create iPart members with different flat pattern characteristics. For example, you can specify different flat pattern orientations or clean up features for each member of the iPart family.

Named Flat Pattern Orientations
Associate different flat pattern orientations with different members of an iPart factory.

Create multiple flat pattern orientations each with its own origin, orientation, and flip direction and save each with a different name. Then designate the right flat pattern orientation for each member in the iPart factory.



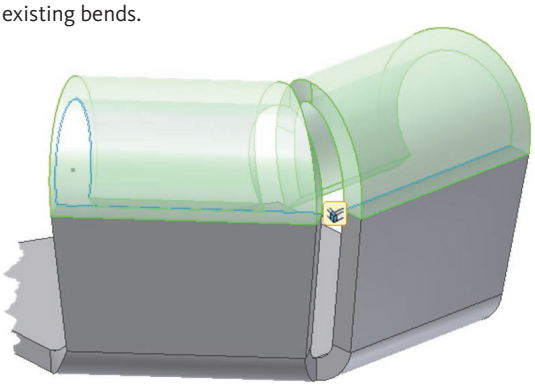
Flat Pattern Drawings
Quickly create flat pattern views of different iPart members irrespective of the saved state of the sheet metal iPart. You can select specific members of sheet metal iParts – or the active member – when placing flat pattern views into Inventor drawings. The flat pattern will be drawn based on the named orientation defined in the iPart table.



Quality Improvements

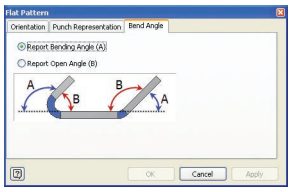
Automatic Mitering of Disconnected Edges

Create mitered multi-edge planar flanges with the Flange and Contour Flange tools – including cases where the flanges are separated by seams or even pre-existing bends.



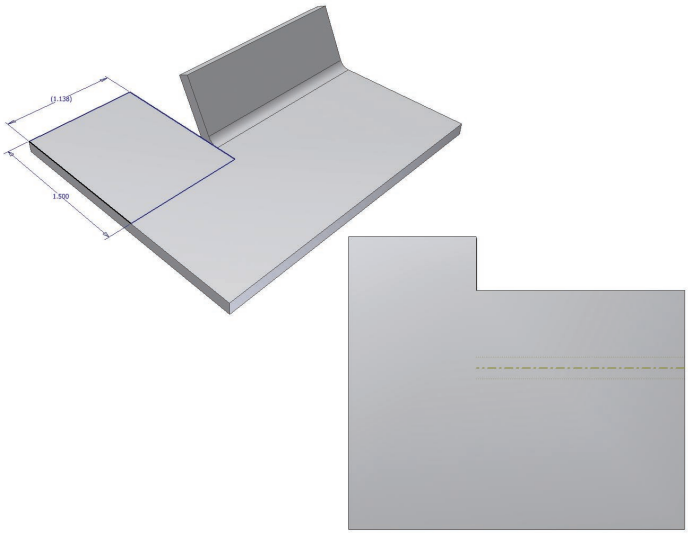
Bend Angle Definition

Change the way bend angles are reported in drawing manager and manufacturing applications that use the API. A new tab appears in the Flat Pattern definition dialog that allows you to select either the bending angle or the complementary open angle.



ASM Simple Tearing

Model faces orthogonal to bend plates without a separation notch.

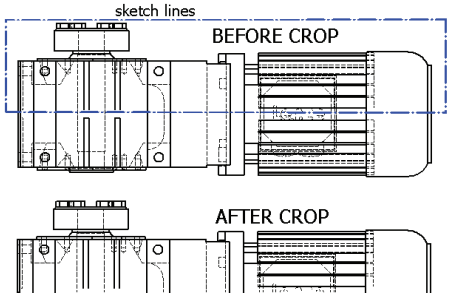


Drawing Manager

Enhancements to the Drawing Manager deliver increased productivity to simplify and streamline the process of producing the drawings required by manufacturing teams.

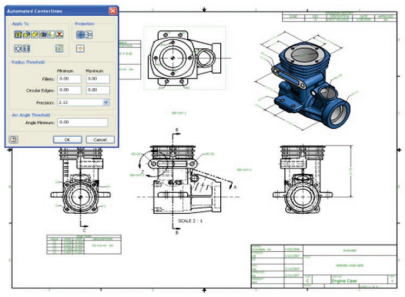
Crop View

Quickly crop views in the drawing when you need to show one side of a symmetrical part or where you only need to show a small part of a larger part or assembly. The new Crop View tool supports rectangular, circular, and polygonal crop windows and avoids the need to use the breakout tools. To use polygonal crop windows, simply create a view sketch and draw the required polygon. The crop sketch supports sketch constraints so you can define the position of the crop window in relation to geometry in the view.



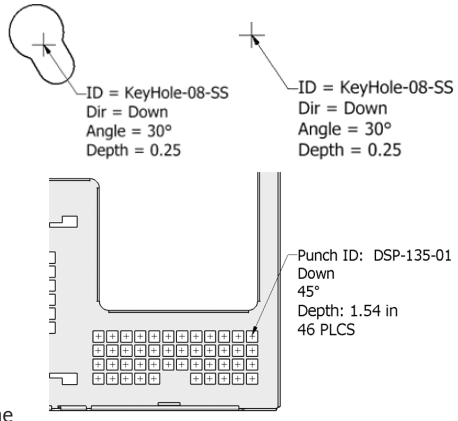
Automated Centerlines

Select one or more views in the browser – or use the layout filters and window select – and use the Automated Centerlines tool to create centerlines automatically on multiple views. The Automated Centerline tool provides options to create centerlines for any combination of holes, fillets, cylindrical, and revolve features, as well as sheet metal bends and punch tools.



Punch Notes

Add punch notes to sheet metal drawings when you don't need a complete punch table. The punch note provides a quick and easy way to document punched shapes. And because the data is extracted from the punch ID, depth, direction, and angle parameters of the punch feature in the 3D model, punch notes will update whenever the punch feature is modified.



The appearance and format of the punch note is controlled through a new Punch Note style, and you can also include a <QTYNOTE> token to display the number of like punches in the pattern or in the part.

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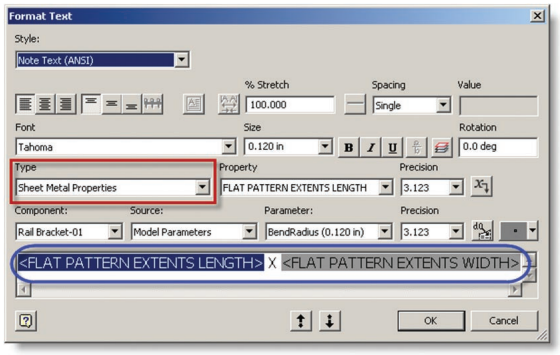
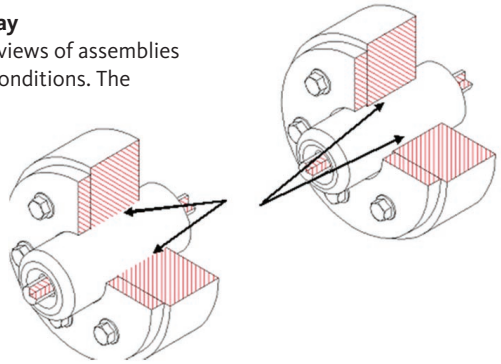
Flat Pattern Extent Properties
A new sheet metal property type has been added which includes the length and width extents of flat patterns for use in drawing manager and manufacturing applications that use the API.



Center of Gravity
Display and use the gravity center mark in drawings. Select the new Center Of Gravity option from the browser context menu of the component referenced by the view. Selecting this option causes the center of gravity of the component to be calculated and a center mark placed at the corresponding location in the drawing view. The COG calculation respects Positional Representations, iPart/iAssembly member state, as well as flat vs. folded sheet metal states.

Interference Edge Display
Create accurate drawing views of assemblies containing interference conditions. The new Display Interference option in the View dialog Display Option tab turns on calculation of interference edges so you no longer need to draw these in afterwards with the sketch tools. Both Visible Interference Edge and Hidden Interference Edge are exposed in the Object Defaults Style, giving you the ability to map them to different layers.



Superscript and Subscript
Easily create text containing fractions, tolerance stacks, chemical formulas, and numbers in scientific notation. Text objects, including general notes, leaders, and system generated text, such as dimensions and hole notes, now support stacked text. Text stacking follows the AutoCAD paradigm triggered by the / # and ^ characters so you can insert stacked text as you type.

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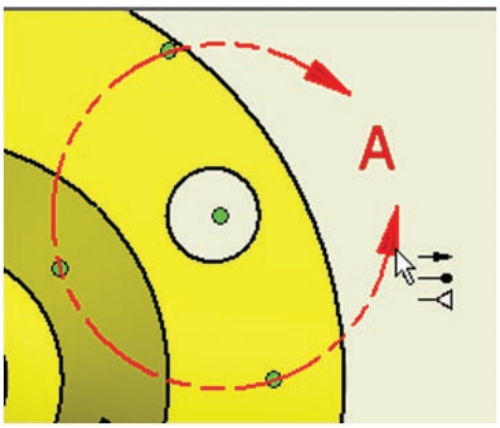
$x\ 10^{-9}$

Parts List Filters
Save valuable time when creating parts lists by controlling the content of parts lists without the need to suppress individual rows. New Parts List filters – accessible from both the Parts List Style and the parts list itself – provide extensive control over parts list content. The following filters are provided: Ballooned Items Only, Standard Content, multiple Item Number Ranges, i.e. 1 through 10 and 20 through 30, Purchased Items, and View Representations.

Reorder Prompted Entries
Change the order of prompted entries in sketch symbol's new Reorder Prompted Entries command located on the context menu of the sketched symbol's browser icon. Eliminates the need to delete and recreate prompted entries when you need to change the order.

View Annotation Style
Use the new View Annotation Style to easily modify the format of view annotation such as section lines and detail view boundaries. The View Annotation Style provides increased control over Section, Auxiliary, and Detail views using graphical menus to simplify selection of the desired style. Additional fields in the style allow users to modify the extension line length, the style and size of the terminator, or arrowhead, and the text style for the label.

Terminators, or arrowheads, used in Section, Auxiliary, and Detail views now behave the same as dimension arrowheads – move the cursor over the green dot at the base of the symbol and double click when you see the symbol cursor to select a different terminator style.



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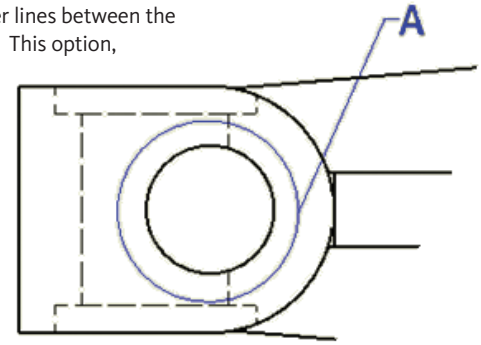
Leader Line Option for Detail Views
Create detail views with leader lines between the detail boundary and the label. This option, which works with the non-ANSI style of detail, is located on the context menu of the detail view annotation. After selecting this option, you can drag the label away from the detail view boundary and a leader line is created automatically.



Enhanced Default Standard Style
Enhanced Options for View Label Defaults
The View Preferences in the Default Standard style have been enhanced giving users control over the content and formatting used when a view label is created. You can control default parameters for each of the different view types so your view labels are created exactly the way you need them. Includes control over label visibility and the label format. You can insert model properties in to the label and control the position of the label relative to the view. A new Constrain to View Border option provides predictably control.

Exclude Character Lists
A new Character Exclude control, located on the General tab of the Default Standard style, allows users to call out which characters are to be excluded during the creation of view names and annotations (tables and tags) associated with revisions, holes, or bends.

Default Thread Representation
New option allows users to control the display representation for sectioned threads and tapped holes.

Rename Sketch Browser Entry
You can now give descriptive names to sketch nodes in the browser by double clicking on the node's icon and typing the new name. This capability works with view level sketches, sheet level sketches, and sketch symbol instances.

Improved Plot Resolution for Shaded Views
Create high resolution plots containing shaded views even when you don't have the 3D part or assembly model. Inventor drawing files now maintain high resolution bit maps of rendered views to improve plot fidelity.

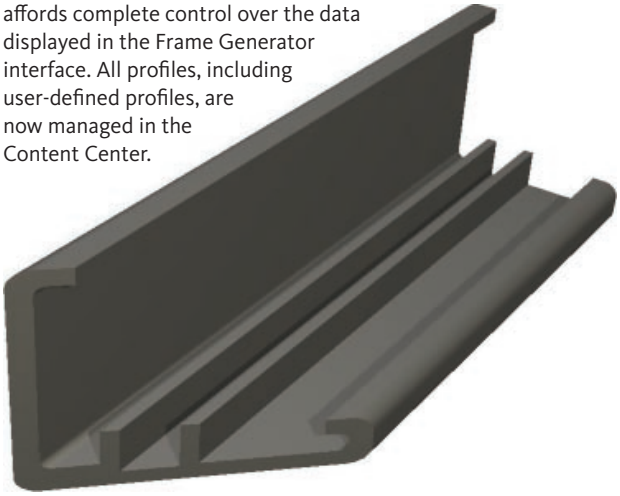
K factor Display in Bend Tables and Bend Notes
The bend information reported to Drawing Manager or via the API now contains KFactor information. The Bend Note tool has been updated to allow users to add this additional information if desired.

ESKD Drafting Standard Compliance
- Wrap additional dimension text underneath a dimension line.
- Terminate leader on corner of feature control frame.
- Two additional GD&T symbols – axis intersection and parallel profile.
- Closed arrowhead symbols for circular and total runout.

Frame Generator

Many companies and individuals require use of custom profiles. In Inventor 2009 software, users can publish their profiles to the Content Center for use in Frame Generator. A new Frame Shape Authoring tool has been developed to facilitate the publishing operations. The tool has the capability to prepare the custom profile for Frame Generator use. The feature affords complete control over the data displayed in the Frame Generator interface. This feature gives the user the necessary tools to add any custom profile to the Frame Generator. Profiles such as aluminum extrusions, dimensional lumber, profiled wood, rails, and miniature tubing to name a few.

User Defined Sections
Inventor 2009 addresses the needs of the many companies and individuals who use custom profiles. Examples of common uses include custom aluminum extrusions, dimensional lumber, profiled wood, rails, and miniature tubing. Inventor 2009 provides a new Frame Shape Authoring tool for defining and publishing user-defined profiles. The authoring tool affords complete control over the data displayed in the Frame Generator interface. All profiles, including user-defined profiles, are now managed in the Content Center.



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Frame Generator – Multi-select
Save time by inserting multiple frame elements in a single operation. Standard Inventor multi-select options, including window select, chain select, and multi-select using ‹ctrl›, allow you to select and populate multiple elements. You can also work with the entire contents of a sketch by selecting the sketch browser node.

Performance and Capacity

The Inventor user community continues to increase the scope and size of projects designed with Inventor. Customers building large industrial equipment and complex transportation systems are already managing assemblies with tens of thousands of parts.

Enhancements in Inventor 2009 provide additional scalability that simplifies working with very large assemblies, extending the performance envelope of Inventor. Customers can now leverage the full power of 64 bit computers and approach even larger projects knowing that they will be able to work with very large part counts.

Native 64 Bit Support
Take full advantage of your investment in 64 bit computers[2]. Inventor 2009 ships with 32 bit and 64 bit implementations. The installer will automatically install[3] the 64 bit version when it detects a 64 bit version of Windows XP or Windows Vista. With this version, you no longer have to worry about the 3 gigabyte barrier allowing you to open and load much larger assemblies.

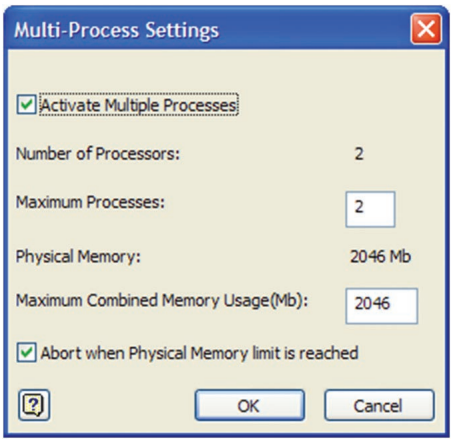
If you install on the 64 bit version of Windows Vista you will be able to install significantly more RAM. The exact amount depends on the design of your motherboard.

Windows Vista Edition	Addressable RAM
Home Basic	8 GB
Home Premium	16 GB
Ultimate	128 GB
Business	128 GB
Enterprise	128 GB

Capacity Meter – Display Physical Memory Option
See at a glance how much physical memory is being used. With the huge increase in virtual memory space available on 64 bit computers, users are more interested in seeing how much physical memory is being used. On 64 bit systems, the capacity meter shows physical memory, or RAM, utilization. You can also select this mode on 32 bit systems from the toolbar tab of the Customize dialog.

Optimization for Pentium 4 Instruction Set
Go faster with Inventor 2009. The Inventor 2009 has been optimized to take advantage of the SSE2 extended instruction sets supported on Pentium 4, AMD Athlon 64 and AMD Opteron processors[4]. While results will vary depending on the operation, you can expect an overall performance improvement in the range of 5 to 15% from this. The Inventor 2009 will not install on computers that do not support SSE2. Several utilities are available on the internet that report CPUID including supported instructions sets.

Task Scheduler – Multi Process Support
Complete batch jobs in less time by taking advantage of multi-process support in the Inventor Task Scheduler. You can run up to sixteen processes at the same time. Set the number of processes and the amount of memory to be used using the Multi-Process Settings control. The task scheduler will launch processes until the specified memory limit is reached.



Footprint, performance, Add-in start-up
Reductions in add-in size have reduced the overall memory footprint as well as the application start up time.

[2] Requires 64 bit operating system: Windows x64 Edition or a 64 bit version of Windows Vista
[3] 64 bit and 32 bit deployments are separate
[4] or later processor variants

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3D Navigation

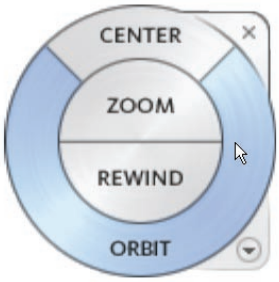
View Cube

Get more control of default views in the modeling and drawing environments. The View cube replaces the glass cube with a small cube located in the corner of the screen to provide faster access to standard views. The view cube has active regions – edges, corners, and faces that give single click access to the corresponding views.

When adding base views to a drawing sheet, you can select the new From Model option to place the view with alignment defined by the front plane defined by the View Cube.

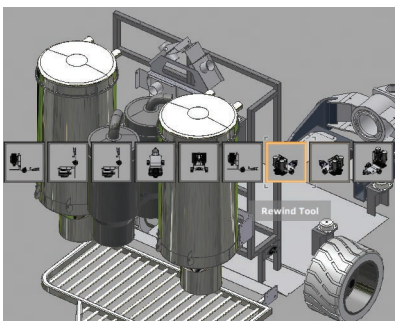
Steering Wheel

Achieve better control of pan, zoom, and rotate, especially when working on laptops, without the use of mice or other pointing devices. The new Steering Wheel provides navigation flexibility and travels through the graphics area attached to the cursor to streamline interaction with the user interface. The Steering Wheel offers many different levels and types of control over model and drawing navigation, as well as providing consistency with other Autodesk applications.



Rewind Tool

Quickly return to previous views. Select the Rewind area on the Steering Wheel and scroll back through the filmstrip to view and select previous views.



Constrained Orbit

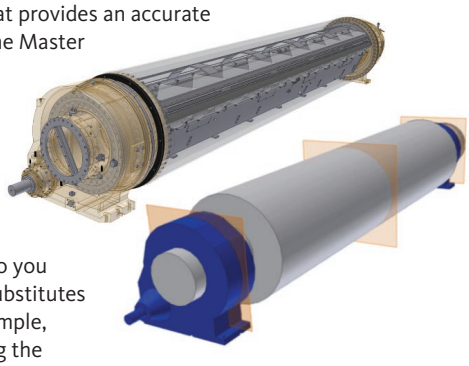
Rotate models as if they were located on a turntable. The existing Orbit tool has been renamed to Free Orbit and is joined by the new Constrained Orbit tool. Use this tool to quickly and easily constrict rotation to the vertical axis.

Assembly Design

Using assembly substitutes, you have the advantages of lighter assemblies that still preserve accurate mass properties, including center of gravity, assembly constraints, and bill of materials data.

Assembly Substitutes

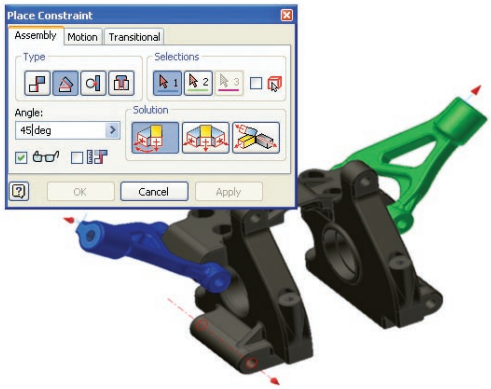
Use a substitute part to represent an assembly when you don't need access to the full assembly representation but still need to see how components and sub-assemblies interact. Substitute parts can now be called out directly from the Level of Detail folder so you can quickly switch between the substitute and any other LoD representation. Use a simple part when you want to hide a lot of detail or use the Derived Assembly tool to generate a substitute that provides an accurate lightweight representation of the Master assembly. The Derived Part tool has a new Reduced Memory mode so substitutes based on derived parts consume even less memory.



Level of Detail supports multiple assembly substitutes so you can create different assembly substitutes for different workflows. For example, create one substitute containing the information required for a digital mockup analysis and a second to define the level of detail needed so you can quickly create large layout drawings.

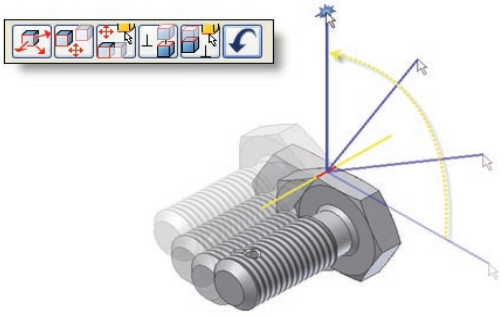
Improved Angle Constraint

Control the behavior of movable joints – such as pins and hinges – by using the new Explicit Reference Vector option in the Angle Constraint to define an unambiguous reference direction.



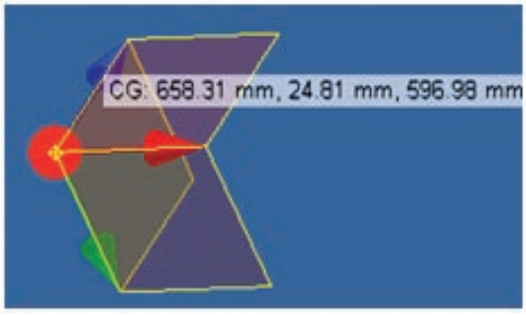
Technical What's New

GripSnap Move and Rotate
Easily explore different configurations during the early design definition phase. New in Inventor 2009, the GripSnap tool combines flexibility, ease of use, and precision in one easy to use tool. GripSnaps supplement the existing assembly constraints allowing you to quickly move and rotate components. Simply select the components, work geometry, or sub-assemblies that you want to move. The possible translational and rotational options for the selected entity are displayed automatically in a pop-up ribbon bar.



AutoDrop - Multiple Insert Option
Reduce the time required to place multiple bolts or screws. There is a new button in the AutoDrop toolbar to enable insertion of multiple fasteners in a single operation. When this option is selected, AutoDrop searches for and populates all holes that match the diameter of the original selection.

Center of Gravity
Include the Center of Gravity in your design process by using the new workpoint and orthogonal workplanes associated with the Center of Gravity glyph.



Place Component Orientation
Quickly place hundreds of components into assemblies without redefining the orientation at each step. Place Component and paste operations now use the orientation from the last instance in the browser and apply this to the next insertion, reducing the need to reorient individual components.

Enhanced Drive Constraint
Use the Drive Constraint tool to identify and observe precise collision information without increasing the number of steps. The Drive Constraint tool now stops at the precise point of collision rather than the nearest step and displays the corresponding constraint angle in the Drive Constraint dialog. You can now use Contact Sets to define which parts should be included in the collision solve to improve performance.

Explicit iMate Matching
Quickly place multiple occurrences of a component using all matching iMate names between that component and other components in the assembly. A new Match All iMates option in the Place Component dialog applies iMates between the placed component and first available component in the assembly with matching iMates.

Part Design

Algorithmic advances and continued investment in the 3D geometry kernel that drives Inventor enables Inventor 2009 to generate well behaved appropriate geometric solutions in a wider range of complex surface conditions. These result in increased stability when modeling advanced shapes and significantly reduce the time required to get the geometry you want.

Shell, Boundary Patch, and Fillet
The Shell, Boundary Patch, and Fillet tools have been enhanced to provide additional modeling power needed for plastic and cast part design. The Shell tool has a new option which allows the selection of a subset of tangent faces, providing more control over which faces are removed by the shell operation. This increases Shell tool capability for advanced shapes and provides greater modeling flexibility. Boundary Patch is now more robust when tangency is applied, and the Fillet tool has a new Minimal setback option which generates smoother transitions between intersecting fillets.

Extrude to a point or vertex
Improve workflows requiring interference extrusions or when creating parts from multiple profiles on different planes. The To option in the Extrude tool can be used with a sketch point, a workpoint, or a vertex.

Technical What's New

Sketch Productivity
The sketch environment receives additional attention to incorporate enhancement suggestion from the user community. Enhancements were selected on the basis of their ability to streamline the user experience and provide improved feedback during sketching tasks.

The principle enhancements are:
- On-Curve Coincident Constraints in 3D Sketches
- Select boundary geometry for Trim or Extend operations
- Streamlined display of constraint icons
- Improved Sketch Edit Tools
- Degrees of Freedom Display
- Create Geometry by Importing Points
- Constraint Inference and Persistence
- Additional 2D alignment options

Geometry Aligned Text
Emboss or extrude circular text patterns using sketch text. Use the new Geometry-Text tool to create text aligned to arcs, circles and line segments with full format control. The resulting text will update when the geometry is modified.

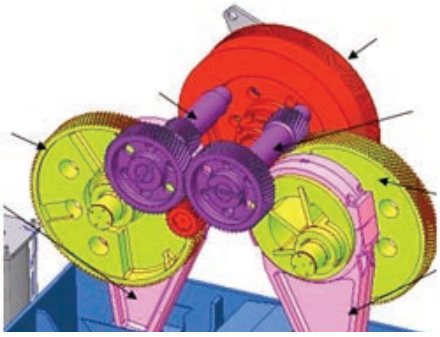
Custom iProperty Formatting
Take advantage of iProperties to populate the Bill of Materials and parts lists with fully formatted parameters such as length and width. Using the new Custom Property Format in the Parameters table, you can now set custom formats to control units, formats, and precision. You can also suppress the units and leading and trailing spaces. There is an option to apply the format settings to all Exported Parameters of the same type in this part file.

Edit iProperty Expressions
Easily modify custom iProperty expressions. A new edit function symbol appears next to fields containing expressions. Use the symbol to access the edit mode and edit the expression directly.

Dynamic Simulation

Numerous enhancements to the dynamic simulation environment make the tools more accessible and easier to use. With additional automation of default properties, users unfamiliar with all facets of simulation conditions can now conduct trial simulations without expert knowledge, while seasoned users can enter a new simulation more efficiently and validate their digital prototypes very early in the design process.

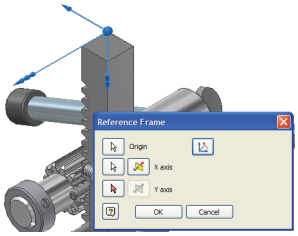
Spur Gears
Use dynamic simulation to analyze the performance of mechanisms containing spur gears. Use the design accelerator to create spur gears, and the dynamic simulation environment will automatically insert a rolling joint and initialize the efficiency, helix angle, and pressure angle parameters.



Output Grapher – Precise Events
Quickly determine the precise time of contact, or impact events.

Precise information about contact events are now displayed as a specific time step in the Output Grapher.

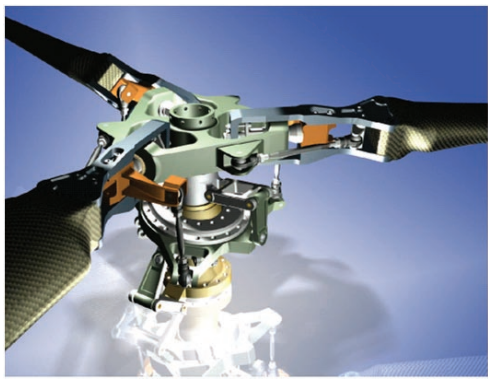
Output Grapher – Multiple Reference Frames
Examine forces and moments in different reference frames. Create new reference frames using the Add Reference Frame button in the Output Grapher toolbar, then select the desired origin point and the X and Y reference directions. Reference frames are displayed in a new Reference Frames folder in the browser panel of the Output Grapher.



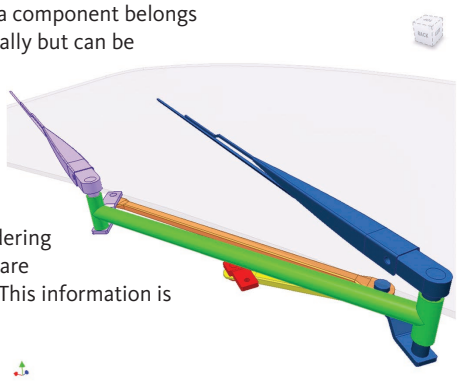
Technical What's New

Output Grapher – Traces
Create additional traces without re-running the simulation. Dynamic Simulation now supports the addition of traces using the cached simulation data. Traces are now managed by the Output Grapher. To add a new trace after running the simulation, just specify the point you want to trace and the cached simulation is used to draw the trace without re-running the simulation.

Output to Inventor Studio
Increase the realism of Studio animations by rendering the output of dynamic simulations. Use the Create Studio Animation command and all of the simulation parameters are automatically transferred to the Studio environment. There is a new default animation called Dynamic_Simulation and a new parameter, Simulation_timeline parameter, for use with the Animate Parameters command.



Improved Color Control
Assign different colors to different mobile groups to make it easier to see what group a component belongs to. Colors are assigned automatically but can be overridden using color styles overrides. You can now override component color within Dynamic Simulation including colors created using Color Mobile groups. Grounded components are shown in a glass rendering to emphasize which components are grounded and which are moving. This information is stored in a separate Design View.



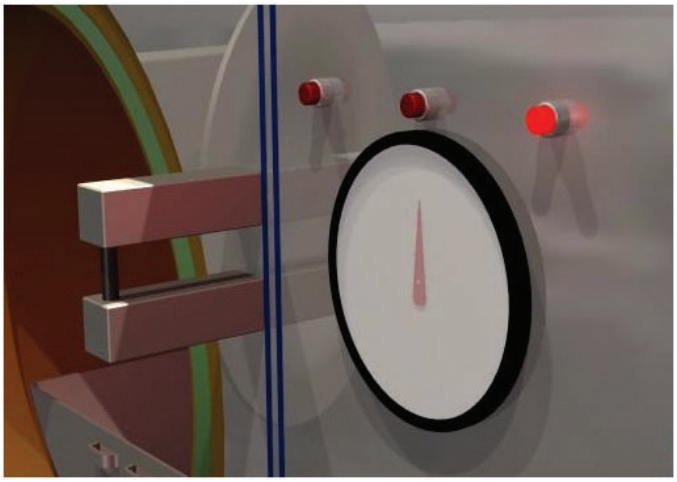
User Interface
- New commands in the Output Grapher toolbar for managing Traces, Reference Frames, and Precise Events.
- Pan and zoom output graphs using the middle mouse wheel.
- Pan and zoom input graphs using the middle mouse wheel.
- Improved display of graph scales when changing zoom factor.
- Input Grapher reference selector relocated for ease of use.

Output Variables
All Output Variables have been renamed to make them easier to read. For example, "ls" changed to Extent_length and "nb_cp" changed to Num_points. A complete list of the variable changes is included in the Help system, under the topic Output Grapher variable names.

Inventor Studio

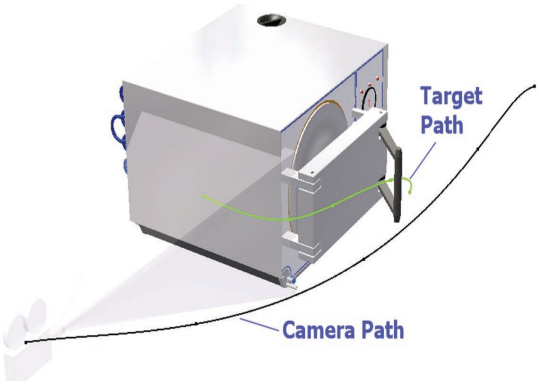
More and more users of Inventor are using Inventor Studio to create realistic renderings of their designs. Inventor 2009 includes many enhancements to its state-of-art rendering and animation tool so design engineers can maximize the realism and effectiveness when communicating with customers and other decision makers, by creating high quality photo-realistic renderings and animations.

Animated Lights
Increase the realism of renderings by adding animated lights to produce effects including back lighting, product lighting, and device lights such as warning lights and on/off indicator lights. Simply define the time position in the animation timeline and define the light's animated state at that time. Many of the light parameters, such as on/off, position, target, color, and intensity, can be animated.

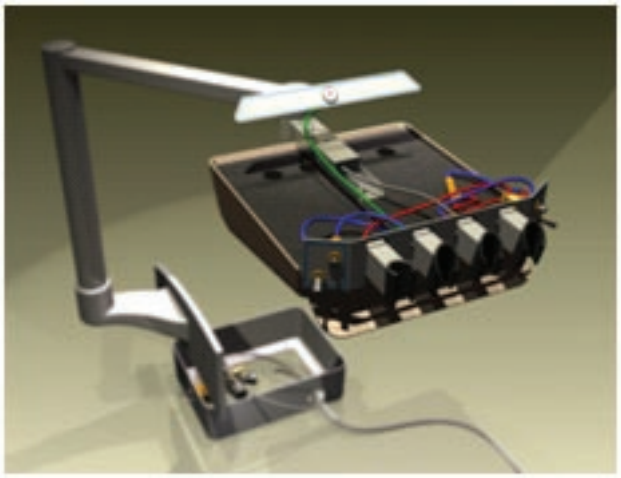


Technical What's New

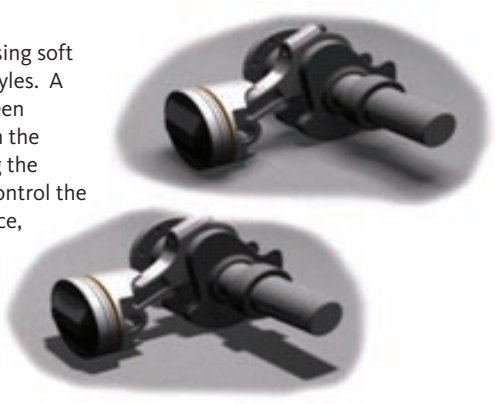
User Defined Camera Paths
Quickly create sophisticated camera movements such as fly-bys and walk-throughs to add realism and maximize the effectiveness of animated renderings. The new camera path animation tools are quick and easy to use. Simply create 2D or 3D splines to define paths for both the camera position and camera target, associate them to the camera and then use the path edit handles to specify the start and end points along each path. The camera will follow the path while pointed at the location defined by the target path.



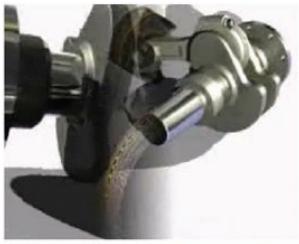
Depth of Field Control
Create images with increased realism by controlling the depth of field on each camera. The depth of field control simulates a real-world camera with focal plane and f-stop settings. Or you can define the near and far focus planes in relation to the model. With a broad depth of field, all or nearly all of a scene is in focus. With a narrow depth of field, only objects within a certain distance from the camera are in focus.

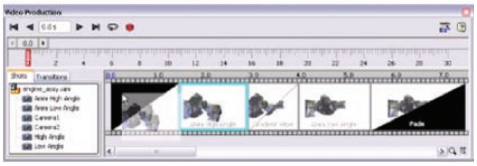


Soft Shadows
Control dispersion when using soft shadows in your lighting styles. A new light parameter has been added to the Shadow tab in the lighting styles. By changing the Light Parameter, you can control the dispersion of light and hence, the quality of the shadow.

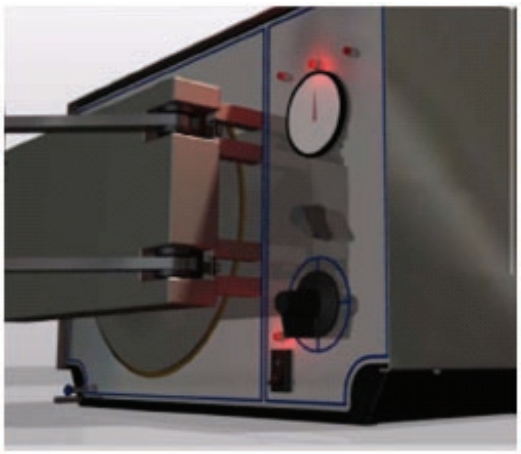


Video Producer
Enhance the creative quality of animated output by adding composited camera shots to your animations. Use existing cameras or add new ones, animated or still, to add video productions to the Productions folder in the scene browser. Simply drag cameras and fades to the Video Producer timeline and render the composited shots to produce stunning visual output.





Local Lights
Simulate lighted components such as panel and switch lighting to improve realism. Local lights move with the component when animated, and their parameters are respected and reflected when rendering images or animations. Local lights can be created from an existing light or as a completely new light and local light parameters can be animated.



Technical What's New

Lighting Style Origin
Reposition Lighting Styles to reuse a set of lights with different areas or components of a large assembly. Lighting styles have a new Position that represents the center point of a sphere that encompasses all the lights in the style. Simply orient the light using the orientation tools, and enter a new XYZ values to move the Lighting Style to a new location. The Lighting style position can also be modified using the 3D Move/Rotate command.

Suppress All/Multiple Constraints
Quickly set up the model for unconstrained animation by suppressing all or top level constraints with just a few steps. Right click an assembly and click "Select Top Constraints" selects only the constraints working on the assembly and not those of the sub-components. Then Right click and you can add the selection to Animation Favorites, or suppress them all at once.

Import/Export for Studio Styles
Easily share surface, lighting, and scene styles using the Export and Import controls in the Style dialogs. Styles are saved with the standard Inventor .styxml file extension. Exported styles can be distributed with deployment images, or passed around between users based upon your style control processes.

Interoperability

Interoperability is a major theme for Inventor 2009 because in today's world of integrated supply chains, designers increasingly need to work with 3D data from customers and suppliers. Inventor 2009 includes the native translators for Inventor that were released on Autodesk Labs during 2007.

Native Translators
Streamline projects that require opening files from vendors or customers in native formats. Promote collaboration with clients and manufacturing teams that need 3D design data in other file formats. Native translators support direct data exchange between Autodesk Inventor and UGS, SolidWorks® or Pro/ENGINEER® by: importing and exporting Parasolid®, importing UG-NX™, importing SolidWorks, importing and exporting GRANITE, and importing Pro/E.

Import Assemblies as Single Part
Create single part files when opening STEP, IGES or SAT assembly files. Use single part files when inserting models of standard or purchased components to simplify the model and increase performance.

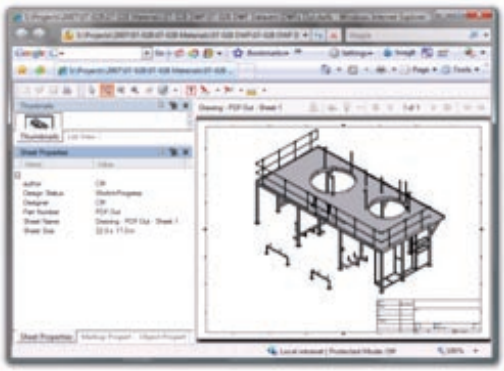
A new checkbox called Import Assembly as a Single Part is located in the File Open Options dialog to select this behavior. With the option active, all solid bodies encountered in the input file are created as a single base object and all surfaces, whether from the same part or different parts, are placed into one composite. Any wireframe geometry encountered in the file will be placed in a group with the same part name. Wires from neutral files are placed into 3D sketches, and points are placed in the construction environment.

Support for Low Tolerance Surfaces
Import and use a wider range of surface data from industrial design applications. Surface data from conceptual design software is often generated with low "mathematical tolerances" that make it difficult and time consuming to incorporate this data in the parametric 3D model. Enhancements to the modeling kernel in Inventor 2009 provide a better user experience when working with imported surface data, particularly when working with lofts, extrude profiles, and sweep rails.

Publishing

The ability to share manufacturing data is a business critical function. Enhancements to DWF™ publishing provide increased performance and better 3D rendering of decals and files containing positional representations. Additional publishing options to help you share your design include support for DWFx, which can be read directly in Internet Explorer V7, and support for PDF.

DWFx Format
Allow third party partners to view drawings without the need for viewers or browser plug-ins. Inventor 2009 supports the new DWFx format, which can be viewed in Internet Explorer Version 7 on both Windows XP and Windows Vista. The DWFx format supports both 2D and 3D data. DWFx files containing 3D data are best viewed using Autodesk Design Review. DWFx is fully compatible with the Markup Manager.



Technical What's New

DWF Performance
Enjoy faster DWF publishing. The Express option now uses the active graphics tessellation rather than recalculating every element. This results in much quicker DWF publishing without compromising quality in most cases.

Improved Visual Fidelity
Provide recipients with more accurate and consistent representation of the parts and assemblies under review. Published files now contain the Inventor perspective view setting and environment map to provide improved visual parity between the DWF output and the Inventor image. Parts and assemblies containing multiple decals are published correctly as DWFs with multiple decals.

Sheet Color
Publish Inventor drawings with sheet colors that match other drawings. Use the Override Sheet Colors option in the DWF publishing tool to precisely control the background color in the resulting DWF file.

Accurate Flexible Components
Publish assembly designs containing flexible components such as wiring or hoses with realistic representation of different assembly positions. Models containing flexible components and positional representations are fully represented in the DWF file, providing more realistic and accurate rendering in each position.

PDF Output
Share design data using the PDF format. Inventor 2009 includes a PDF output to give you more format flexibility. PDF output from an Inventor drawing produces vectors and text. PDF output from a 3D model is rendered as a bitmap image.

Design Accelerators

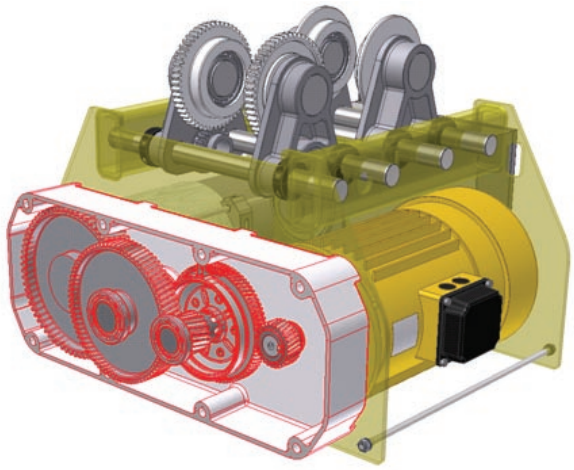
Inventor 2009 reduces the learning curve associated with rapidly designing, analyzing, and creating commonly used machine components based on real-world attributes, such as speed, power, and material properties.

Bolted Connections
Spend less time inserting bolted connections. The bolted connection tool includes the following enhancements:

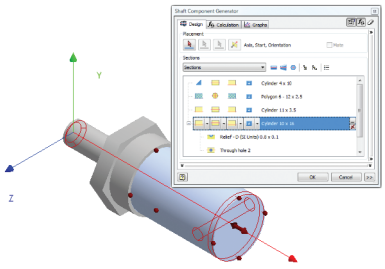
- Select fasteners with different parameters – such as material type – using the table dialog.
- Insert bolted connections into hole patterns based on center points. Simply select the original patterned hole and the specified bolted connection is inserted into all patterned holes at once.
- Insert bolted connections based on points, 3D points, or work points created in sketch environment. Insert bolted connection into all sketched points at once, or unselect points where bolted connections are not required.
- Use the Concentric placement option in the Design tab to insert bolted connections into holes created with Extrude and Revolve commands in the part environment.

Gears
Spend less time designing reduction gears. The gear generators provide better graphical feedback:

- 3D grips added to preview mode to allow fast adjustment of gear dimensions.
- New ANSI/AGMA 2001-D04 strength check added for spur and bevel gears.
- Bevel gear preview now shows helical teeth.
- New flip button was added to flip the Start plane.



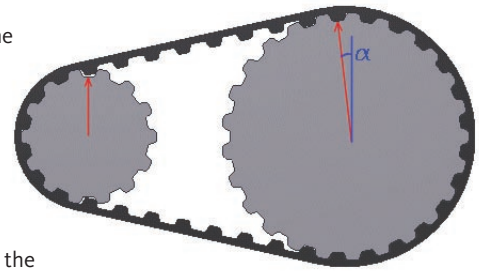
Shafts
Complete the as-manufactured shaft design using the Shaft Design Accelerator. New section features include inner and outer threads, keyway grooves with one rounded end, plain keyway grooves, and special chamfer for n-sided prism. In addition, inner bores can now contain fillets, chamfers, threads, and grooves for inner retaining rings .



Technical What's New

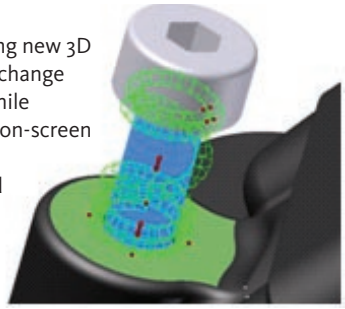
O-rings
Complete the design of shaft and bearing assemblies using the new redesigned O-ring Generator. Select a radial or axial O-ring from Content Center and insert it into the assembly.

Belt Timing
Lock the initial rotation of the desired toothed pulley with selected reference geometry. Use the new options in the More area of the Design to specify additional constraints that define the initial starting position. You can also adjust the initial rotation of each teethed pulley using the tooth grip. The first tooth in the pattern is displayed and the pulley can be rotated with respect to the belt pitch.



Belt Representation
Select the best representation for belts in the assembly model. The existing solid representation is joined with two new options: Sketch, which generates belts as a simple sketch with dash-dot line type, and Detailed (available for Synchronous Belts only), which generates a solid including teeth represented as a patterned iFeature.

Springs
Interact with springs using new 3D grips controls to quickly change the spring parameters while viewing the result in the on-screen preview. Supports spring position, outer, inner and mean spring diameter, wire diameter, and number of coils.



Cams
New commands were added to enable flip start plane for both linear and disc cams. For linear cams, you can now also select a start plane.

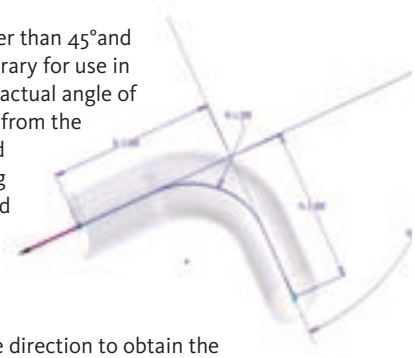
Weld Calculations
Modify the Allowable Stress value in the Joint Material and Properties area of the weld calculators.

Templates – Export and Import
Save time by saving and reusing templates. Export and import design accelerator parameters is supported for bolted connections, shafts, spur gears, worm gears, V-belts, and synchronous belts. The Export template command stores data in the XML format and the Import template supports both XML and CAL (legacy) data.

Tube & Pipe

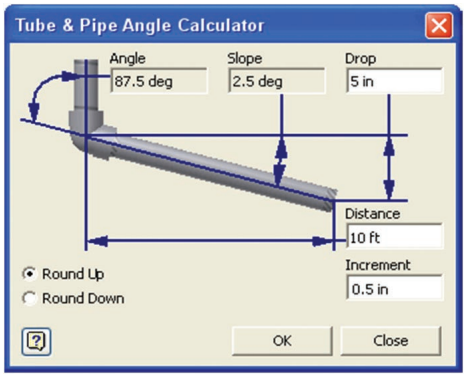
Design process lines that comply with sterility and cleanability standards required for process equipment in the pharmaceutical, food, and personal care industries. It's now easy to incorporate the correct slope into the pipe route including the use of drainable angle fittings that comply with ASME BPE standards.

Custom Fittings Support
Add fittings with angles other than 45°and 90° to the content center library for use in self draining pipe runs. The actual angle of the fitting is now calculated from the part geometry and displayed in the Tube & Pipe Authoring dialog. This angle is also used when defining self draining tube and pipe styles.



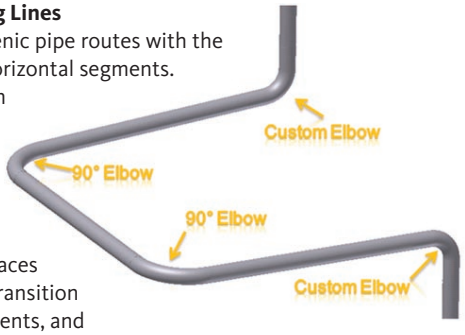
Gravity Vector
Define the vertical reference direction to obtain the correct run-off directions. The new Gravity tool allows you to identify the vertical direction by selecting a face or edge.

Self Draining Tube & Pipe Styles
Define tube and pipe styles to support the design of self draining lines. Select the "self draining" option in the Tube & Pipe Styles Editor and enter the target slope or fall off angle. Or use the angle calculator to compute the angle from a specified drop. In addition to standard 45 and 90 degree elbows, self draining pipe styles include a custom elbow for use with fittings with non-standard angles such as 92 or 91 degrees.

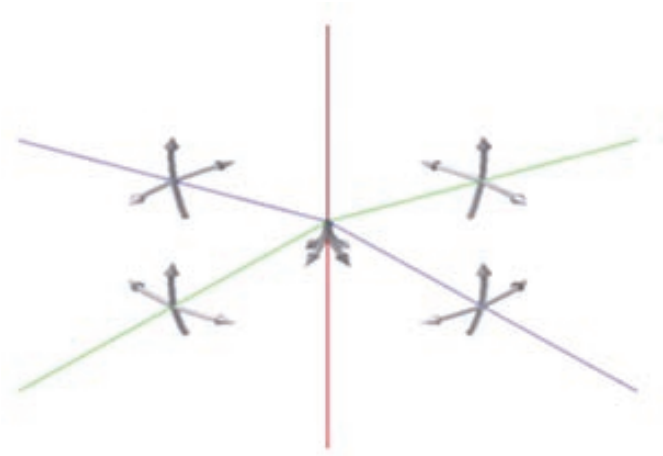


Technical What's New

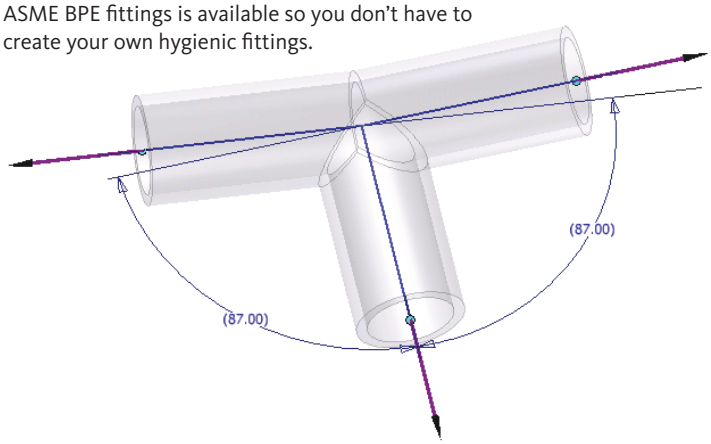
Auto-route for Self Draining Lines
Automatically generate hygienic pipe routes with the correct runoff to eliminate horizontal segments. The Auto-Route tool has been enhanced to support custom angled fittings by applying the slope angle defined in the active Tube and Pipe style to the necessary route segments. It automatically places custom angle elbows at the transition from vertical to sloping segments, and 90° elbows at the transition between two sloping segments. As in the past, the fitting information is determined by the active Tube and Pipe Style.



Enhanced Routing Tool
Quickly select the right run off direction when editing tube and pipe runs. Based on the gravity vector, the Tube & Pipe editor automatically displays the different run-off directions making it extremely easy to select the correct option.



ASME BPE (style) fittings library
Incorporate ASME BPE compliant fittings into hygienic tube and pipe designs. A library of commonly used ASME BPE fittings is available so you don't have to create your own hygienic fittings.



Enhanced Filters for Library Browser
Select the fitting with the correct angle from a library containing multiple fittings with the same nominal angle. The library browser is enhanced with new angle filters that allow you to specify a specific angle or a range of angles. The browser has been reorganized with a vertical selection panel so you can specify and find the right part in less time.

Cable & Harness

Multiple Control Point Modification
Rapidly reposition wire segments and harnesses by editing multiple control points using the improved 3D Move/Rotate tool. Changes include an enhanced path preview so you can clearly see the effect of the changes and an Apply button to support multiple editing of multiple control points. Invoke the tool once and interactively reposition one or more control points until you have the path you need. You can move control points individually or as a group and then select the Apply button to commit the changes.

Defer Update for Individual Wire Harnesses
Enjoy increased performance while editing harnesses with large wire counts. Right click on a harness node and use the new Defer Updates options on the Harness Settings dialog to suspend update length and bill of material updates. Two options are provided: Defer All Updates and Defer Length Updates in the BOM, both of which apply to the active harness. Defer update icons in the browser show which items are in the deferred state. In addition, wires in the deferred state are displayed as zero length in the bill of material to draw attention to their deferred status. Defer update settings for other harnesses can also be accessed and modified from the Harness Settings dialog.

Content Center

Save As for Content (Family)
Create part families with different materials by copying an existing part family with the new Save As command. The Content Center Editor includes a new Save Copy As command located on the toolbar. The new part family can be independent or linked to the parent family so any updates to the parent family are automatically propagated into the new family.

Separate Servers for Content and Vault
Separate the Content Center and Vault servers by installing Autodesk Data Management Server on two computers, and then use one as a Vault server, and the other as a Content server.

Technical What's New

Favorites
Customize the structure and content of the Favorites list. You can now use Copy or cut and then paste folders or components (families or family members) within your Favorites group, or from the Content Center group to your Favorites group. You can also drag and drop folders or components within the Favorites panel.

Vault Client

Spend less time dealing with the Vault and more time working on your design. Enhancements to the Vault client reduce the time required to retrieve files when you are starting work and less time returning files when you've finished. Enhancements to Vault workflows eliminate unnecessary interaction and internal improvements increase the performance of checkin and checkout operations by avoiding unnecessary network traffic.

Single Step Check In
Save time when checking in work into the Vault. The Add Files and Check In commands have been combined to provide a single check in process that automatically adds files that are not already in the Vault. Check In operations are faster due to improvements in DWF publishing and optimizations made in the Check In command itself. In addition, you can now access the Check In command directly from the Vault browser. Note that the Add Files command has been removed.

Direct Access from File Dialogs
Access the Vault directly from frequently used file dialogs. Click on the Vault icons located in these dialogs to work with files from the Vault. The Vault icon is available on the File Open, Place Component, and Derived Part dialogs as well as the Base View dialog in the Drawing Manager environment, and the Create View dialog in the Presentation environment.

Browser Performance
Experience better Vault browser performance. Browser overhead has been reduced by performing updates to the status icons only when the browser is accessed and only for items visible at the top level. Lower level status information is retrieved only when the assembly item is expanded.

Support for LOD Suppression with Open from Vault Operations
Enjoy faster checkouts when working with complex assemblies. When you call out specific Level of Detail or Design View representations in the Open from Vault dialog, Inventor 2009 downloads only those files required for the selected representation, reducing server traffic and increasing checkout speed.

Separate Server Login (Disable single sign-on)
Configure separate servers for Vault and Content Center. You now have the option to specify and log into a separate server for Content Center libraries and the Vault. From the File menu, select Autodesk Data Management Server > Connection Options. Separating content and vault servers provides more options for optimizing performance in large installations.

Technical Documentation

Inventor 2009 extends and improves its range of learning and reference resources to help users maintain skills and quickly make the most of the 3D design environment.

CAD Manager Fundamentals
Find help on setting up and managing Inventor installations to provide and maintain a productive design environment. Refer to the new CAD Manager Fundamentals resource for information on installation, migration, application options settings, configuring styles, best practices, and more. An installation option is provided so you don't have to install the CAD Manager Fundamentals on user's machines.

New Features Workshop
Review the new features in Inventor 2009 software using the enhanced New Features Workshop built into the help system. Similar to the New Features Workshop in AutoCAD and other Autodesk products, the Inventor New Features Workshop provides a graphic-rich introduction to What's New for the release so you can understand what is new at a glance.

Improved Help
Take advantage of in-depth information on how to use Inventor. This release includes improved topical depth and navigation of content throughout the help system. Over forty new animations, five new tutorials, and multiple examples and static images have been added to the content, along with a restructured table of contents and home page for improved navigation. Finally, an informational video guides new users on how to use Help.

Licensing

License Timeout

CAD administrators can now set a maximum idle period for network licenses to optimize license utilization. Licenses that are left unused for longer than this period are returned to the license pool and become available for other users. Sessions reactivated after the timeout period automatically seek a new license from the license server allowing users to return to their work. There is a two hour grace period that covers situations when all licenses are in use. The idle timeout option is disabled by default within the application. Enable timeout monitoring by configure the options file. The minimum timeout period ADLM allows is 900 seconds (15 min).

Installation and Deployment

Installation

Improved Performance

The installer has been tuned to increase performance. The product initialization process now takes place after the product selection page so you don't have to wait for the installer to initialize the install process for products that may not be installed.

Configuration Reminder

A warning message appears if you press the "Install" button and have not invoked the configuration pages.

Reorganized Installer Documentation

Additional and improved installation documentation is accessed from a new button on the main startup page of the setup program.

Return to start button added to last page of installer avoids the need to restart the installer if you need to perform another operation – for example create an additional deployment or reference the documentation.

Adobe® Flash® Player

The Adobe Flash Player is no longer installed by the Inventor installer.

Deployment

Administrators now have the ability to exclude component products from a deployment if the deployment is not configured to deliver the product(s) to workstations.

It is now possible to create and modify deployments on workstations that already have the software installed.

Administrators can apply 32 and/or 64 bit service packs to deployments regardless of the administrator's workstation platform.

The Create and Modify Deployment Shortcuts have been moved to a tools folder below the Administrative Image folder to reduce exposure of this functionality to non administrative users.

Ability to create a 32 bit or 64 bit deployment is no longer determined by the desktop operating system, so you can now create both 32 and 64-bit deployments from a single computer.

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.