Autocad inventor 2013 tutorial pdf

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The screen is divided into three sections: Work Section - Start new files, or configure the default templates. Learn Section - Open interactive tutorials, watch short introductory videos, or view the What's New topics. Extend Section - Start Autodesk Exchange, or explore the cloud with Autodesk 360. A series of short videos designed to help you learn key tasks and workflows is included in the product. You can open the videos from the Welcome screen or the Get Started tab. Inventor has a great new learning environment to help you get started. This environment leads you through tutorials with step by step video, supporting text, and in-canvas call-outs linked to the text. They run inside the Inventor window. You can start the tutorials from the Welcome screen or the Get Started tab. The Create New File dialog box layout is redesigned to provide file information and better organization. Templates display in groups by type. Information about the template, such as units and materials, is displayed. A thumbnail displays for each template type. In-command marking menus are aligned to have consistent placement of the OK, Done, Cancel, and Apply options. In-command marking menus display when you right-click while a command is active. Context-sensitive marking menus are available for sketch objects in Inventor. Select objects, then right-click to display the marking menus are available when you rightclick, or right-click and drag (gesture behavior). A user-defined marking menu is available in each Inventor environment. You must define the Customize dialog box to define the Ctrl+right-click menu. Hold down Ctrl and, right-click to access the customizable marking menu. The Ctrl+right-click menu contains constraint commands in the 2D sketch environments. The overflow menu are defined for Inventor to streamline the interface. The original, longer overflow menus are on by default. Use the Marking Menu tab of the Customize dialog box to access the short overflow menus. The options available in the Customize dialog box to control the display of the overflow are: Full Menu Radial Menu Only Short Menu When you click Extrude or Revolve before you create a sketch, an error message displays, and you can start a new 2D sketch. The default setting for some Application Options is changed. The following options are affected: Sketch tab: Edit dimension when created is on. The following user preferences are removed from the installer, and are now available from within Inventor post-Installation. User Profile - The User Profile options are now available at the bottom of the Application Options dialog box: Click the down-arrow next to Import and select Use AutoCAD Related Settings - The Measurement Units/Drawing Standard default settings have been moved to the Configure Default Template dialog box. Access the Configure Default Template dialog box: Select the File tab, and then click the Configure Default Template button. Part Modification settings have been moved to The Application Options dialog box: Select the Enable part modification from within drawings checkbox to enable or disable. When you start the Create 2D Sketch command, the origin planes display. Pick the edge or face of a plane to begin a new 2D sketch. Use the Two Point Center Rectangle command to create a rectangle defined by a center point and a vertex. Use the Three Point Center Rectangle command to create a rectangle defined by a center point and two points that define the size and direction of the rectangle sides. In the sketch environment, use the Dimension command, Arc Length option to add arc length dimensions. Use the Trim tool to dynamically trim lines and curves. Create or edit a sketch. Then on the ribbon, click Sketch tab - Modify panel - Trim. In the graphics window, hold down the left mouse button, and move the cursor to extend sketch geometry. The 3D Move/Rotate command uses a mini-toolbar to manipulate the X, Y, Z values. The mini-toolbar to manipulate the X, Y, Z values. Mirror command is available in the 3D Sketch environment. Use origin planes, work planes, or flat faces to mirror 3D sketch geometry. These controls, you can customize the line color, line type, and line weight of selected geometry. These controls function the same for 3D sketch geometry as they do for 2D sketch geometry. In the 3D Sketch Line command, you can snap points to a plane. The select does not constrained to the plane. With the line command, you can snap points to a plane. environment uses a mini-toolbar. The mini-toolbar is off by default. To display the mini-toolbar, expand the Draw panel in the 3D Sketch ribbon tab, and click the Precise Input command. Inventor supports two types of splines: interpolation, and control vertex. Interpolation splines and control vertex spline type using the right-click menu. A change in the spline type using the right-click menu. A change in the spline type sometimes causes the loss of constraints and dimensions. Interpolation splines fit the spline type using the right-click menu. type of spline is available in previous releases of Inventor. In a control vertex spline, the selected points define the vertices of the control frame. Control frame defines the shape of the spline. You can constrain and dimension the control frame. vertex splines are available in both 2D sketches and 3D sketches. You can apply tangent, perpendicular, and smooth (G2) constraints anywhere along control vertex spline but not at the end point of the spline. You can now apply a tangent constraint between the spline and the line, and to other geometry types besides lines. Modifying geometry in a 3D sketch works like modifying geometry without entering coordinates. You can modify geometry when a 3D sketch is not active. Drag points to reposition them, and roughly position geometry without entering coordinates. on 2D sketch geometry, or right-click a point and invoke the 3D Move/Rotate command. The changes to a 3D sketch display in a preview while you enter changes to a 3D sketch display in a preview while you commit. Equation curves are useful to model complex geometry, such as gear tooth profiles or sweep paths for hydraulic pumps. Equations into 2D sketches. You can enter explicit equations are supported in 2D and 3D sketches. When working with implicit equations, provide the equations for X, Y, and Z (for 3D sketches) and the range of values to evaluate. The mirror command in the 2D sketch environment has a Self Symmetric about the mirror line. When this option is selected, the mirror command creates a single spline that is symmetric about the mirror line. Some 2D sketch constraints are enhanced: Perpendicular constraints accept arcs. You can apply tangent, perpendicular, and smooth (G2) constraints: Anywhere along control vertex splines, projected geometry, or equationcurves. To the endpoints of interpolation splines. You can add user-defined parameters of True/False or Text type to text in model or drawing sketches, and to drawing notes. To edit sketch text, in the Sketch environment, double-click the text object. Use the Find in Browser. The related sketch highlights in the browser. A new icon is added to the Part browser to indicate fully constrained 2D sketches. In the Assembly environment, the indicator icon updates its appearance when you exit the Sketch environment, it updates interactively as you add or remove constraints. In 2D Sketches, when using the Measure Distance command, you can select a midpoint of a linear segment, and a circular arc. Commands that were previously located on the 2D Status Bar: Snap to Grid Show all Constraints Hide all Constraints Dimension Display Slice Graphics Show All Degrees of Freedom Hide All Degrees of Freedom Commands on the 3D Status Bar: When you edit a part in the context of an assembly, you can project sketch geometry is now associative by default. The sketch and the part are set as Adaptive to keep the geometry associative. On the Assembly tab, an Application Option controls whether projected sketch geometry is associative. The option is called Enable associative sketch geometry projection during in-place modeling. The function to create a box, cylinder, sphere, or torus. Primitives can add or remove material, or create new bodies. The Offset command is added to the Edit Base Solid environment. Offset provides the means to modify radial and planar faces of imported solids without opening Inventor Fusion. You can enter a precise value, press to remove thickness, or pull to add thickness. Offset does not add a new feature to the model. Select multiple part features, right-click, and click Properties. The Feature Properties dialog box with the Feature Properties dialog box with the Feature Appearance to all selected features. In the browser in the part environment, you can move the End of Part marker directly under a selected feature. Right-click the feature and then click Move EOP Marker. Move EOP Marker command is not available when you select multiple features in the browser, or when it is already located under the selected feature. directly to the top, or directly to the end of features in the browser. When the End of Part marker is located within your part features, right-click in the browser, and click Move EOP to Top, or Move EOP to End. In the Parameters dialog box, the state of the display of folders now persists. For example, if you collapse the Model Parameters folder, it remains collapsed the next time you open the dialog box. In parts and assemblies environments, sketch points within a selected area can serve as centers of holes. Click the Hole command. In the Placement area drop-down menu, ensure that From Sketch is selected, and click Centers. Click and drag to select an area. Sketch points within the selected area are used for the centers of the holes. The Shell dialog box is collapsed by default. Offset and angle work planes use direct manipulation and mini-toolbars for creation and edit. This workflow is consistent with other Inventor features. An error glyph is displayed next to the mini-toolbar when a feature cannot be calculated. The error glyph is displayed while the command is active so you can address the error before committing the change. Shell Equation Curves The construction environment is disabled by default in Inventor. Therefore, commands related to the construction environment instead. The construction environment is available in Inventor, when you open legacy parts that contain construction data. Enable the construction environment using the option in the Application Options, Parts tab. You can now create view representations and capture the following display characteristics: Part Appearance Sketch Visibility Work Feature Visibility UCS Visibility Browser Expansion State Zoom Magnification Viewing Angle Components dialog box, use Copy to insert a list of components not suitable for reusing are highlighted in the Mirror Components dialog box. If reused components dialog box, use Copy to insert a list of components dialog box, use Copy to insert a list of components dialog box. suitable for reusing to the clipboard. Then paste the list to a document. When you create derived or shrinkwrap solid body parts, select the Remove All Internal Voids option to fill in all internal void shells. In Frame Generator, you can select geometry to align frames instead of defining a rotation angle. On the Design panel, click Insert Frame, Specify the frame member and itsposition, and check the Align box. Select desired geometries are not selected due to a potential conflict. For example, the frame cannot be aligned to the geometries that belongs to a: Frame that already aligns to the selected frame. End treatment generated by Miter, Notch, Trim To Frame, Trim/Extend, and Shorten (curved or merged beam). Part or assembly that has an assembly constraint with the selected frame directly or indirectly. In assembly constraint with the selected frame directly or indirectly. treatments of an arc curved frame are trimmed following the curve shape. End treatments on curved frame are trimmed through tangent direction. Use these Frame generator tools to edit end treatments of a spline curved frame are trimmed through tangent direction. first level of the menu to the second level for parts and sub-assembly. The functionality is not changed. To open a component drawing, in the Assembly, and then click Component - Open Drawing. In the browser, right-click, and click Delete. Review or refine the selection in a dialog box, and then confirm the sheet deletion. All selected sheets are deleted from the drawing. Select one or more views, right-click, and click Copy. Right-click in the graphics window, and click Paste to place copied views. Hold down CTRL to disable leader line snapping in 15-degree steps. Then place or more the leader node at any point in the drawing. Insert image types BMP, GIF, JPG, PNG, and TIF into a title block. In the Parts List dialog box, click the + sign in front of a parent row, and click Expand All Children. The Base View command selects by default the last active model document as the source for the placed view. If the last active model document is closed, no model is automatically selected. New in-canvas appearance editing tools make it easier to modify textures right in the scene, and provide immediate feedback. in-canvas with real-time updates. In the mini toolbar you can modify the appearance color, change the assigned appearance, and rotate the appearance texture map in-canvas. Contextual feedback helps to determine the necessary modifications. The texture manipulator works on the selection of a feature or face The automated adjust appearance process reduces the level of libraries include: Autodesk shared libraries for Material and Color Styles, for a more consistent design and viewing experience. The installed libraries include: Autodesk Inventor Material Library - A consolidated subset of the Autodesk Material Library with a focus on manufacturing related materials. Autodesk Material properties, whether viewed in Inventor or Revit. Autodesk Appearances in other library - A compliment of appearances accessed by multiple products and suites. Using the library browsers, you can: Search for specific materials or appearances to the document. Access materials or appearances in other libraries. Create custom libraries to suit your manufacturing requirements. Access additional libraries, and manage custom libraries, and manage custom libraries, and repearance properties, and so on. Each product uses assets according to the product requirements. The project file specifies the material and appearance libraries for that you want immediately accessible for that session. There are multiple access points for tools to work with materials and appearance libraries for that session. creating and modifying materials. Appearance Browser - Use for creating and modifying appearances. Adjust command - Use for removing appearance overrides. Assigns the specified material to the selection set. Materials and appearances are assets that define the composition and look of your design. Materials are grouped in categories, such as metal, wood, ceramic, and so on, with subtypes for each category. For example, the metal category might include subtypes of aluminum, steel, titanum, and so on. Appearances are also grouped in categories, such as color, patterns, texture images, bump maps, and other properties. Each type has properties unique to the type. The combined properties provide a unique appearance. A collection of dialog boxes, a browser, an editor, and an asset browser are available for managing materials and appearances. The browsers provide access where you can manage libraries and their content. Create custom libraries and populate them with assets. A toolbar at the base of the dialog box provides library access and management tools, and includes the means to duplicate and modify assets. You can create custom libraries. The editors present information specific to the material or appearance asset, the corresponding assets display in the browser. You can replace material and appearance assets for any document material. The common use of texture maps to define the appearance of components, and modeling with textures displayed, adds visual complexity and increase efficiency, a pair of controls are added to the View tab - Appearance panel. Using Textures Off, you can quickly override the display of textures while modeling, and redisplay them for visualization. You can set these controls to persist with each model using Document settings, or with each session using Application Option settings. The display regardless of the texture setting. Autodesk 360 online services are available for all subscription customers. Integrated access to online services is added to Autodesk Inventor 2013, and all subscription customers have access to online document storage. You can use your secure login to access to online resource where you can browse and purchase Apps (including product-specific content such as models, training materials, and eBooks) - and is available to several Autodesk Products. You can download some items for free, and some for a price. Exchange App Manager that is included with the respective Autodesk product. In the stress analysis environment, new functionality performs the analysis of thin wall components. You find the bodies within your model that meet the thin component criteria. Then you simplify the geometry to generate midsurfaces or offsets, and perform the analysis. On the Prepare panel, use Find Thin Bodies to find and highlight bodies that meet the shell features criteria. Then, use Midsurface inspects the selected bodies and automatically builds a median surface based on the geometry. Shell features are considered in the analysis for such surfaces. Offset creates a shell feature from selected faces. Specify the thickness, and distance is calculated as thickness/2. You can perform an automatic face chain selection. An API for the Dynamic Simulation environment opens up some very exciting co-simulation workflows with other simulation tools. The API allows you to control the magnitude of any load at each time step and it also allows you to extract resultant information like position and acceleration, enabling you to interface with external control system simulations. To learn more about the API, open Programming Help on the Autodesk Inventor Help menu, click Community Resources and then Programming Resources. Multi-core machines see enhancements relative to mesh creation and analyses. All available machine cores are now used for both mesh generation and the solution phase of your simulation resulting in faster performance. An expand section is added to the Mold Process settings dialog box to create a virtual sprue. With a virtual sprue, you can perform a mold fill analysis without creating a mold base or an actual sprue. To create the virtual sprue, select a runner sketch, and specify the virtual sprue drop-down menu, and select Merge Core and Cavity into Mold Base Plates. An option is added to include surface bodies from the source model when you derive the plastic part. This option provides greater flexibility for creating complex hole patches and runoff surfaces. Select the surfaces to include surface bodies from the Sprue Bushing dialog box provides greater flexibility in setting the location. You can locate the sprue on a runner sketch by selecting the sketch and specifying a ratio value. The ratio value adjusts the position of the sprue along the sketch and specifying a ratio value. performance and capacity of mold assemblies: Multi-body in the moldable part creates an insert for improved performance, capacity, and robustness. The derived mold part uses Reduced Memory Mode when the plastic part is placed in the mold assembly. Additional enhancements are: The Delete option is supported in the context menu of both Create Patching Surface and Create Runoff Surface. When you place a plastic part, the color style of the original part is inherited. The Specify UCS option is added to the Part Shrinkage. You can save time during the design process by independently updating the core/cavity without updating the core/cavity without updating the core/cavity or mold environment. A point cloud is a set of vertices in a three-dimensional coordinate system, defined by X, Y, and Z coordinates. The vertices typically represent the external surface of an object. You can: Attach one or more point clouds to both part and assembly files. Move, rotate, scale, crop, and uncrop a point cloud. If you use the Autodesk Factory Design Suite, snap factory assets directly to points in the point cloud. The Import Multiple Solid Parts as Assembly option is no longer available. Files that contain multiple solid parts are imported as Inventor multi-body parts. affects import options for JT, Parasolid, IGES, STEP, NX, and SolidWorks files. For SAT files, the options are Import as a Single Part, and Import/Export: Version 9.5 Granite Import/Export: Version 7.0 NX Parasolid Import/Export: Version 24 Rhino Solidworks SAT Export: Version 7.0 only (version 4-6 are no longer supported for export) Run the new Purge task to purge styles and materials for multiple files in one operation. Use this task when you transition to using a style library with your design project. A purge operation: Removes every unused style from all of the Autodesk Inventor files (parts, assemblies, assemblies). drawings) that you specify for processing. Removes styles that you no longer need from legacy documents. Often reduces the memory requirements for working with large assemblies. Any styles that you do not save in another document or a style library are permanently lost in purging. The CAD Manager and IT Manager Resource is dedicated to CAD Managers and IT Managers. Links are provided on this centralized location to get you to the information you will find links to topics on: Data Migration Language Packs and Language Switching Product Support resources There is a new Data Migration resource that details the factors to consider when deciding how to manage your data from one release to the next. Determining if and what to migrate may depend upon any combination of the following: the type of data you create, the quantity of files you manage, and whether you are using a data management system (such as Vault). The new Data Migration resource includes detailed step-by-step procedures for migrating design and data files. To determine the best migration process for each type. A language pack is a mechanism used by some Autodesk products to provide another language for the user interface and the documentation. You can now install additional standalone language switching functionality is now supported for: Autodesk Inventor Professional R2013 Autodesk Inventor LT R2013 AutoCAD R2013 AutoCAD R2013 When you launch Autodesk Inventor or Autodesk Inventor LT by a language specific Start menu shortcut, that language for Inventor. The online Wiki Help is now the default Help for Autodesk Inventor and LT. If you are online and access the Help, the online Wiki Help displays. If you need to work offline and need access to the product help, you can download the Help locally to your computer. When you are offline, the local Help will display. CAD Managers and IT Managers and wizard. Well, there it is. Keep checking back as I will be adding some short videos on how the new functionality works in the coming weeks. You can also follow me on alot of the social networks included in the following post. Different Ways to Follow Inventor Topix Page 2 Home Inventor What's New AutoCAD What's New Lynn Allen AutoCAD Videos Links Inventor Topix Store Contact Me ▼

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