

I'm not robot!

Unwrap Command: The new Unwrap command is used to calculate flattened areas of faces that cannot be calculated using either the “Unfold” or “Create Flat Pattern” commands in the sheet metal environment. The command creates a merged or unmerged resultant surface that can be used in downstream processes or used to create detail drawings. An optional “Create Dedicated View Representation” is available. Solid Sweep: The ability to sweep a solid along a path to create various Boolean operations is a new addition to the Sweep command. Along with many options this command will make creation of complex geometry much easier. Relationship Enhancements: Sketch blocks consumed by part features in Relationship dialog box are no longer displayed. Support for Multiple Face Selection: You can now select multiple cylindrical and conical faces of extruded holes in multiple solid bodies using the Thread command. Property Panel Based Commands: The number of commands using Property Panels has increased including Extrude, Revolve, and Sweep. Enhanced Property Panels to supply better command functionality and user interface enhancements. Many can assign Presets, configure panel icons, apply and create icons, sketch and feature navigation and increased tooltips. Loop and Region Detection: Enhanced choice of profiles and regions. For example, in the Extrude command below, picking the regions enclosed by sketches will select just those areas and selecting the actual sketch profile will select the entire area within the sketch profile. Graphic Presets: Three new Graphic Presets are available to give a quick look and feel for your designs. Each one designed to give various degrees of visual display quality and performance. User Interface Refresh: Refreshed Inventor browser and ribbon icons to update the display. A new “Light Theme” is available as a color scheme. Define a section texture display to use a saved bitmap image of your choice. Content Center Enhancements: Perform searches in the Content Center using the two standard Windows wildcard characters “?” and “\*”. Categories folders that do not have subfolders no longer show a “+” in front of them. Inventor Read Only Mode: Automatic installation of Inventor Read Only Mode along with the application and appears in the application folder structure. Use this mode to view and print Inventor files without consuming a license on a trial basis. Once the trial period has expired, it can be used in the place of Inventor View (Inventor Read Only Mode does not expire). Inventor View will still be available as a separate installation if desired. Multi-Monitor Support: With Multi-Monitor Support you can drag document tabs to a separate monitor in a side-by-side or full screen mode including the browser and all palettes. The Ribbon cannot be moved and will appear on the main display only and the browser will always show details of the active document tab regardless of the monitor location. Document windows appear on the Windows taskbar as separate screens. Migration Utility Enhancement: An Enhanced Migration Utility, used to migrate Inventor settings to a new installed version, includes: Property Panel settings such as preset and last used settings. Migrate property panel preset lists. Numerous 3rd Party Translators and Performance Enhancements: Upgrades to many 3rd party translators to support new versions. A wide variety of Inventor processes have been the focus of performance upgrades. Assembly Constraints: When using the Assembly Mate constraint between part axes, Inventor will automatically select the solution that causes the least rotation in the display, but manual override is always available. The Assembly Constraint dialog box values are no longer persistent in the same Inventor session. Express Mode Enhancements: The following commands are now available in Express Mode: Drive Constraint Create and edit 2D sketches Work feature selection and visibility control Visibility of shared and unconsumed sketches Selection Priority: Select Sketch Features Skeletal modeling workflows are now available in Express Mode. Assembly Sketch: The Eclipse command has been moved to the Circle tool group to provide consistent access in both Part and Assembly environments. Productivity Tools: In the Create Derived Substitutes dialog box, you can right click on an assembly row and then select to remove it from the operation. Tube & Pipe: The Application Options -> File -> File Naming Defaults dialog box now includes a Tube & Pipes tab. Using this tab you can specify how Tube & Pipe creates files and folders. You can use the attribute drop down lists to configure file and folder names. If desired, you can drop the AIP folder entirely. Design Accelerator/Shaft Generator: The Shaft Generator dialog box now includes a toggle that allows you to specify how files and folders are created for shaft generator components. Mirror Components/Sheet Metal Enhancement: The Mirror Component command now recognizes a sheet metal part and lets you link the sheet metal style of the source component with the mirrored copy. Support for Threads in Assemblies: You can now create Thread features in both assembly and weld assembly models. Frame Generator: Use the new Insert End Cap command to close structural members ends. You can specify position, profile shape, thickness and other properties using a new properties panel. This command uses a custom Content Center library which must be loaded before use. You can also customize this library to suit your needs. An End Cap entry has been added to the Frame Generator File Naming Default options. The Notch command now has a property panel and the ability to notch more than one member at a time. Custom notches can now be specified and configured from the properties panel. Note: Custom Profile must be configured and saved in a Custom Content Center Library. A new Perpendicular Cut possibility is available to create precise intersections. In-Canvas Orientation Tools allow you to directly manipulate a frame member during placement or during a frame member editing process. The Lengthen/Shorten command now has a new property panel and offers you the ability to enter asymmetrical offset values. The Miter Command has a new property panel and allows you to do multiple miter joint selection at the same time. The Trim/Extend to Face command has a new choice which allows you to specify the side of a structural member you want to keep. The following image shows how this works. The Remove End Treatments command now allows the choice of multiple members at the same time using a new properties panel. This year we celebrate Inventor’s 20 year anniversary with the release of Inventor 2020. This release delivers a host of enhancements to help you solve complex product design challenges and get more done in less time. Access the Inventor 2020 release from your Autodesk Account. Inventor 2020 delivers: Enhanced User Interface (UI) and Workflows: Streamlined part modeling, smart sketch profile detection, multi-monitor app frame, and a modernized look and feel. Customer Driven Improvements: Solid sweep, frame design productivity, complex surface unwrap and flattening, read-only mode, settings presets, and several more from Inventor Ideas that came directly from you. Continued Delivery of Professional Grade Functionality: Faster import, patterning, navigation, Tube & Pipe route edits, as well as expanded Express Mode functionality. Multiple Fixes Improving Stability and Quality: Click here for a detailed list of these improvements. Read below for more details on what’s new. UI Enhancements Inventor 2020 delivers a fresh new Light Theme interface that enhances functionality and improves productivity. The Light Theme includes an overall visual refresh for lighting styles, icon refresh throughout Inventor, graphic presets for changing multiple model view settings, multi-monitor support, and extended migration enhancements for moving from past versions to Inventor 2020. To learn more, see User Interface Enhancements, Multi-Monitor, and Migration Enhancements. Command UI and Productivity Enhancements Starting with the Measure Command in Inventor 2018, and the Hole Command in Inventor 2019, additional commands are refreshed with a new property panel UI that includes functionality and workflow improvements. Inventor 2020 extends the property panel UI to the Extrude, Revolve, Sweep and Thread commands to enhance functionality and increase productivity. To learn more, see Feature Productivity and Usability Enhancements. Performance Improvements Inventor customers are constantly pushing the software with larger and more complex designs. To help our customers, we are continually looking for ways to improve Inventor performance. In Inventor 2020, you’ll experience improved performance with assembly, part, drawing, and AnyCAD workflows. To learn more, see Performance Improvements and General Assembly Enhancements. Design Enhancements Inventor 2020 continues to deliver new functionality to core design commands and workflows. Enhancements are made to sketching and Content Center to improve overall productivity. Sweep is improved by adding Solid Sweep functionality for removing and adding swept geometry by sweeping a 3D tool body. An Inventor® Ideas submission The new Unwrap command is introduced to allow you to get a flat surface from any set of continuous faces. Unwrap lets you flatten nearly any shape into a surface that can be saved or exported in different formats. An Inventor® Ideas submission To learn more, see Feature Productivity, Workflow, and UI Enhancements and Content Center Enhancements. Significant improvements are made to Frame Generator. Many of the Frame Generator commands now use the property panel UI. Install the Custom Content library to use the new End Cap command added with Inventor 2020. Notch and Insert Frame functionality are improved. An Inventor® Ideas submission To learn more, see Frame Generator Enhancements. Translation and Interoperability AnyCAD for Fusion 360, which was released in 2018.2 as a Tech Preview, is now fully released as a feature for Inventor and Fusion 360 subscribers. AnyCAD for Fusion 360 is for customers who need to share data between Fusion 360 and Inventor for collaboration, generative design, electro-mechanical, and other workflows. To learn more, see To Import Fusion 360 Files as and AnyCAD Reference Model and To Work with AnyCAD in Fusion 360. Many of the 3rd party supported versions are updated for file translation with Inventor 2020. To learn more, see Translation Enhancements The Inventor community continues to provide feedback and requests through Inventor Ideas and Feedback Community. Improvements labeled with An Inventor® Ideas submission in the What’s New indicate new functionality delivered as a direct result of your requests. The customer driven improvements in Inventor 2020 include the Ideas indicated above, as well as: The sheet metal style of the source component can now be included with the Mirror Component for sheet metal parts. An Inventor® Ideas submission To learn more, see General Enhancements in Assemblies. Joints can now be added to Slot geometry. An Inventor® Ideas submission To learn more, see General Enhancements in Assemblies. Sketch blocks consumed by features are now reported in the Relationships dialog box in parts. To learn more, see General Enhancements in Parts. Inventor 2020 can now be launched in Inventor Read-only Mode to view native Inventor files. Inventor Read-only mode does not require licensing. To learn more, see Inventor Read Only Mode. Your feedback is very important to the Inventor team. We value your feedback and product improvement requests. Please join the following communities to provide your feedback to the Inventor team. Feedback Community Inventor Ideas Inventor Forums AutoCAD Mechanical drawing templates are no longer included in an Inventor installation. Download AutoCAD Mechanical template files, from Inventor 2020 Language Packs.

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