



ULTIMATE GUIDE TO INVENTOR

APPLIED SOFTWARE, GRAITEC GROUP

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CHAPTER 1

[Autodesk Inventor](#) is professional level CAD software used for product design and engineering, including 3D mechanical design, simulation, tool creation, and design communication. With Inventor, you can save time and expense when making products. The parametric Inventor 3D digital model enables the design to be validated on-the-fly under real-world conditions for form, fit and function without needing to build a prototype. Simulation of motion, deflection and stress allows you to optimize the product design. Direct editing and advanced surface modeling features enable the creation of intelligent product components.

Whether you use Inventor to animate, invent or visualize your world, you'll find that connections between products is a foundation of Inventor use. For more fabulous features, see the blog post "[3 Reasons Why You May Catch the Inventor Bug.](#)"

The top two dozen best features and benefits of Inventor:

- Better innovation.
- Less time bringing the product to market.
- Associative link enables the use of existing DWGs.
- Repurpose neutral 3D CAD file formats.
- Parametric and free-form modeling.
- Shape generator.
- Motion simulation.
- Stress analysis.
- Illustration and animation tools.
- Optimize designs without the need for prototypes.
- Cloud-based design reviews.
- Tools for routing rigid pipes, flexible hoses and bent tubes.
- Advanced tools for visualizing, animating and illustrating.
- All necessary fabrication instructions sent directly to the 3D model.
- Automated and associative bills of materials.
- Generate production-ready drawings without the need for a separate CAD system.
- Automate mass customization.

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- Rules-based design and automation.
- Large assembly and drawing performance.
- Specialized tools for sheet metal and frame design.
- Remove unnecessary geometry from a design.
- Optimize loading conditions and target weight.
- Open, connected workflow for non-native Inventor data.
- “AnyCAD” connection with nine CAD/CAM software programs – no translation required.

INVENTOR BASICS

Inventor includes a complete set of professional-grade design and engineering tools. For one of the most popular articles on Inventor Applied Software, Graitec Group has ever posted, read “[5 Common Problems for New Inventor Users.](#)”

SYSTEM REQUIREMENTS FOR INVENTOR 2023

| | |
|-------------------------|---|
| Operating system | Microsoft Windows 10 64-bit Microsoft Windows 11 64-bit |
| CPU | 4 or more cores, 3.0 GHz or greater recommended. Minimum 2.5 GHz or greater. |
| Memory | 32 GB RAM or more recommended. Minimum 16 GB RAM for assemblies with 500 or fewer parts. |
| Video display | 3840 x 2160 (4K) recommended. Preferred scaling: 100%, 125%, 150% or 200%. Minimum 1280 x 1024. |

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|------------------------|---|
| Video adapter | Recommended 4GB GPU with 106 GB/S bandwidth and DirectX 11 compliant. Minimum 1 GB GPU with 29 GB/S bandwidth and DirectX 11 compliant. |
| Disk space | 40 GB required for installer and full installation. |
| Spreadsheet | Full local installation of Microsoft Excel 2016 or later for workflows that create and edit spreadsheets. Inventor workflows that read or export spreadsheet data do not require Excel. (Excel Starter, Open Office and browser-based Office 365 applications are not supported.) |
| Pointing device | MS-Mouse compliant. Productivity: 3DConnexion SpaceMouse, driver version 10.7.0 or later. |
| Browser | Google Chrome or equivalent. |
| .NET Framework | .NET Framework version 4.8 or later. Windows Updates enabled for installation. |
| Connectivity | Internet connection for web install with Autodesk Desktop App, Autodesk collaboration functionality, web downloads, and licensing. Autodesk Network License Manager supports Windows Server 2019 and 2016, Windows 10, Windows 7 SP1. |

See the [Autodesk Knowledge Network](#) for system requirements when developing complex models, complex mold assemblies and large assemblies of more than 1,000 parts. Autodesk also provides a list

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where you can search for tested and [certified graphics hardware](#).

INVENTOR “SUPER COMPUTER” SPECIFICATIONS

- Processor (CPU):
 - Intel Core i9 12900K
 - Intel Core i7 12700K
 - If you want a higher core count that might help with rendering, consider the AMD Ryzen 9 5950X with 16 cores
- Hard drive: 1 TB Solid State hard drive
- Memory: With the amount of rendering you do, at a minimum you should have 64GB of RAM; try for 128GB of RAM if possible.
- Video card: NVIDIA RTX A6000 48 GB



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Go beyond 3D with a single digital model created in Inventor. Integrate design data from all phases of your product's development. In the short video "[Digital Prototyping – Autodesk Inventor](#)," see how an accurate 3D digital prototype can be created to design, visualize and simulate products without the expense of producing an actual physical prototype.

Using Inventor, you get professional-grade mechanical design solutions. But if you've never seen what Inventor can do, you may have some misconceptions. Did you know Inventor can be used to design jewelry displays? See what else Inventor can design, render and simulate in the blog post, "[5 Things Everyone Gets Wrong About Inventor](#)."

Autodesk Inventor has three particular enticements that may tempt you to take a closer look. Learn about useful features and get a first-hand look at the way Inventor works in the on-demand webinar "[Catching the Inventor Bug](#)." Jason Miles explains using Inventor to invent, animate and visualize parts and products that make our world go around. As you will learn, Autodesk went "big" with Inventor. If you've been considering switching over to Inventor, this may be the time to let it transform your manufacturing process.

The best way to get the best designs is to iterate and explore many design options - but it's expensive and time-consuming. Inventor reduces design costs and time to market, by helping engineers and designers to explore more design options, with greater speed, while producing a product that satisfies form, fit and function. Inventor's Generative Design tools are powered by artificial intelligence, which explores thousands of possibilities for innovative designs. The designs are not invented by artificial intelligence, rather they are human designs **refined** using artificial intelligence. Learn about the workflows in the post: "[How generative design fits today's manufacturing processes](#)."

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INVENTOR UPGRADE NOTES

A vast majority of software installs go without a hitch. Inventor is no exception. Occasionally though, installs can appear to go smoothly, and it's not until you try to use certain functionality that you realize there are some problems. Troubleshooting, uninstalling, and reinstalling are the standard approach to fixing broken installations and most probable causes. However, workarounds are sometimes the best or only solution, as described in this older but still useful [Inventor 2017 workaround](#).

During an Applied Software, Gritec Group Inventor Fundamentals class, the instructor was demonstrating the new DWG Underlay functionality in Inventor when he made a discovery. Learn what happened when illustrating the hole and thread note tools in Inventor: "[Inventor Hole Note Errors](#)."

2021 UPGRADE

Find out about Inventor 2021 in the on-demand webinar, "[What's New with Inventor 2021](#)," hosted by Dave Morse. Leave behind repetitive tasks. Experience the improvements in drawing creation. Plus, benefit from these features:

- AnyCAD for Revit
- Drawing automation
- Frame generator enhancements
- Better user experience

2022 UPGRADE

Following is just a sampling of the new and [updated features](#) to speed up manufacturing design workflows in Inventor 2022:

- Model States, a powerful new workflow, was added.
- New options allow you to quickly identify under-constrained

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- components.
- A new assembly level Simplify command replaced the Shrinkwrap command.
 - Instance, a powerful new functionality, lets you assign properties to individual component instances.
 - To synchronize a substitute with the source assembly, you no longer need to open a substitute part and use the context menu command to check for updates.
 - The ISOGEN Output command was improved.
 - New parts workflows were added to increase productivity.
 - Shaded drawing views were updated to use the lighting style from the model.
 - Options were added for Design View that allow extracting Camera View and 3D Annotations.
 - Centerlines and center marks with extended lines now break when intersecting with dimensions.
 - A specialized workflow enables model simplification as part of the RVT export process.
 - A new workflow was added making it easier to share Inventor part files with Fusion Team and Fusion 360.
 - QIF 3.0 support was added so you can export a Quality Information Framework file with PMI data.
 - JT export support was added to Task Scheduler, enabling export of a neutral 3D CAD file containing PMI data.
 - Enhancements were made to iLogic.
 - Enhancements were made to the user interface.

2023 UPGRADE

Following is just a sampling of the new and updated features to speed up manufacturing design workflows in Inventor 2023:

Assemblies

- **Bill of Materials:** A new BOM Settings option was introduced that allows you to control zero quantity visibility and item

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- number sequencing.
- **Constraint Workflows:** You can now suppress a constraint directly when editing it.
- **Express Mode:** An update to component visibility has improved the performance of Express Mode.
- **Model States:** When a substitute model state is active, four commands are disabled in the ribbon and two are disabled in the context menu. Dynamic Simulation displays a message that the substitute model state has no moving bodies. In addition, Frame Analysis displays a message that you need to switch to a non-substitute model state.
- **Tube and Pipe:** Last used values are preserved after you close Route panel in the Properties dialog, including Auto-route, Auto-dimensions, Auto-constraints, and Convert Auto-route to Sketch.
- **Simplify Command:** The way Features are recognized is clearer for the delineation between additive versus subtractive features, including Pocket for subtractive features like sweeps and Emboss for positive protrusions.

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working together for highest efficiency?
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Business Process engagement.**

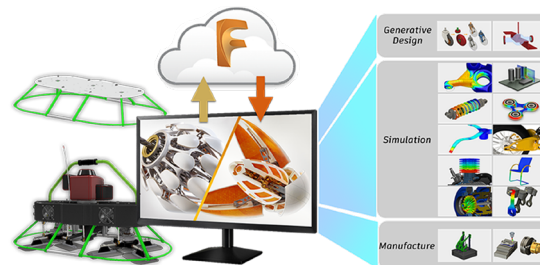
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DRAWINGS

- **Drawing Annotation:** Weld annotations are improved, with a new “field flag direction” option. The latest ISO 2553-2019 standard is used for Edge Weld and Butt Weld Symbols. A Stake Weld symbol has been added for EN ISO 2553:2017. The Surface Finish Collection symbol is updated for the latest EN ISO 1302:2002 standard. All surface texture reference types can be included on a sheet.
- **Drawing Views:** You can change the detail view boundary shape from circular to rectangular by using Edit Detail Properties in the detail view. The command can be accessed from the view context menu in either the browser or the sheet. In addition, you can now copy and move revision tables across drawing sheets.

INTEROPERABILITY

- **Inventor to Fusion 360:** When using Fusion 360 tools you have the option to start the Fusion 360 workspace for the command you choose. The Modeling command (formerly Send to Fusion) takes you to the Fusion 360 Design workspace. The new commands provide workflows into additional Fusion 360 workspaces: Generative Design, Simulation and Manufacturing. You select the Fusion 360 tool you want to use and the components you want to send to Fusion 360. The Inventor components are derived into a single IPT, copied to Fusion Team and opened in Fusion 360 in the appropriate workspace.



Inventor and Fusion 360 interoperability; image: Autodesk.com

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- **Data Exchanges:** Data Exchange for Inventor makes it easy to share specified subsets of Revit data with stakeholders.

PARTS

- **3D Annotation and Model-Based Definition:** A Datum Target command has been added to the General Annotation panel. With it you can establish datums on irregular surfaces. Datum Target has been added to Styles and can be used in Tolerance Features. The Tolerance Advisor has been updated to support ASME 2009 and ISO 2012.
- **Features:** Enhancements include Fillets, relationships, Extrude and Revolve.
- **Mark Command:** The Mark command enables creation of laser marking, etching and engraving features. Mark export properties have been added to the Style and Standards editor.
- **Sheet Metal Parts:** The Face, Contour Flange and Flange commands are now supported by two enhanced options – “Show extended names” and “Display extended information after feature node name in browser.”
- **Tolerance and Parameter:** Part and assembly model states now support unique tolerance settings for each state. Individual model states respect tolerance settings. Tolerance Parameters are accessible in an Excel spreadsheet for model states. Model states can also configure user-defined Text parameters and true/false (Boolean) parameters.

PRESENTATIONS

- Any storyboard can be renamed during creation or when editing a presentation.
- A new Frame Rate option has been added to the Video Resolution settings.

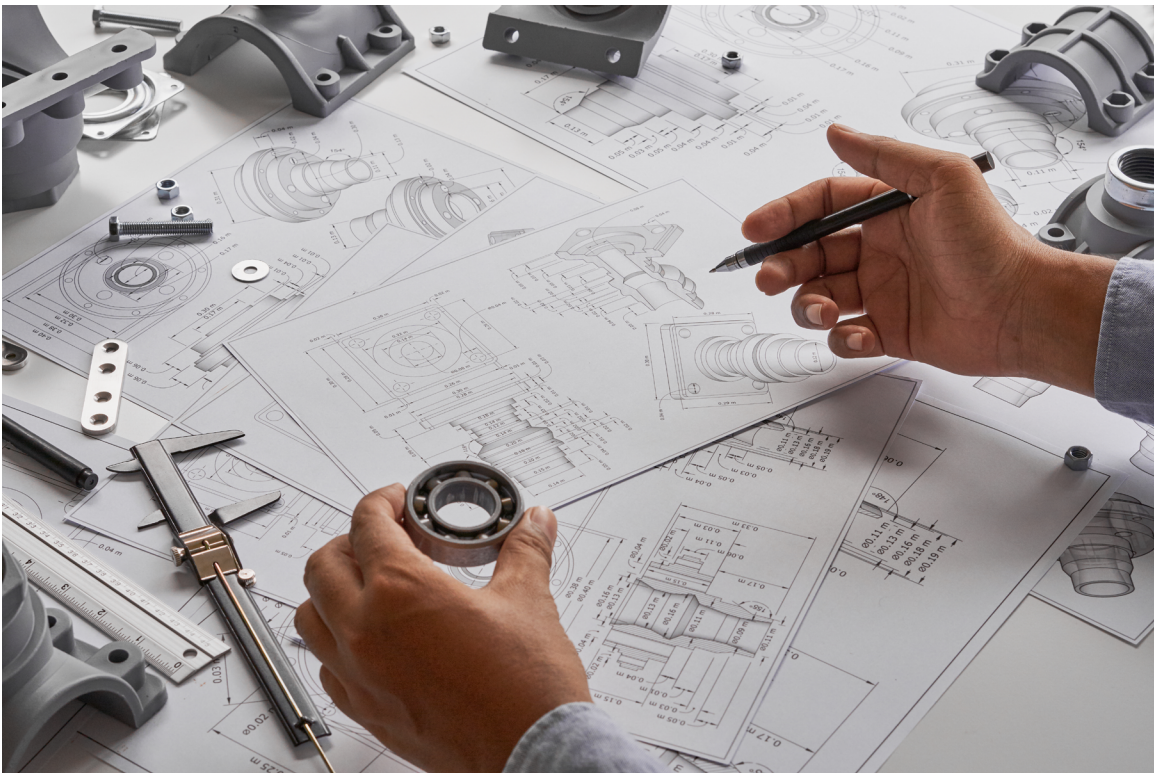
CHAPTER 3

SKETCH

- Standard iProperties and Custom iProperties can now be used in Part model sketch.

GENERAL

- iLogic: You can now add a custom command for an external rule or global form to the ribbon. The filters “iLogic Rules” and “iLogic Forms” are added to the Customize dialog.



CHAPTER 4

INVENTOR COMPANIONS AND ADD-INS

ILOGIC

Using iLogic, a model designer can control parameter and attribute values for the design. ILogic embeds rules as objects directly into a drawing, part or assembly document. So, the knowledge is stored in the document in the same way geometry is stored.

With [iLogic](#) you can standardize and automate design processes when configuring your virtual products, saving time and boosting productivity. As described in “[The Ultimate Glossary of iLogic Terms](#),” each of the iLogic rule functions is accessed by expanding the appropriate node under the System tab in the Snippets area of Inventor.

ILogic rules can make use of custom parameter types available in [Autodesk Inventor](#), like text, true/false and multi-value lists. These parameter types allow rules that involve more than numeric input values. The new possibilities really are compelling. Learn about iLogic and parameter files in the blog article, “[6 Enhancements That Make iLogic Irresistible](#).”

Using iLogic Design Copy is a great way to copy an assembly that’s set up to drive parts from an assembly by [using iLogic code](#). As you’ll see from the description, the process to perform iLogic Design Copy is fairly straightforward.

In another on-demand webinar about iLogic, Jason Miles demonstrates what enables rules-driven design and how it provides a simple way to capture and reuse your work. Using iLogic, you can standardize and automate your design processes and configure your virtual products. Learn about the useful features of iLogic in “[Inventor iLogic Customization 101](#).”

VAULT

If your team is challenged trying to find past revisions of digital prototypes for printing or viewing; if team members prematurely or accidentally use the wrong manufacturing drawings for production; if industry and company standards are not being adhered to; if batch printing of the correct set of drawings is tedious, then Autodesk Vault could help with your team-based design. Vault is an Inventor add-in, and you can learn about the approaches for using Vault at: “[Best Vault Strategies.](#)”

Access the on-demand webinar [Vault Pro and Inventor Migrations 101](#) to learn how to automate pipe support design within Inventor. Whether or not you’re experienced in creating pipe supports, you’ll learn helpful tips and tricks, including:

- How to import a pipe support from Inventor.
- How to create a pipe support report from Plant 3D.
- A demo of intelligent pipe support in Inventor with adjustable parameters.
- A demo of support drawing with BOM.
- How to import modified support into Plant 3D.

[Vault](#) has options that enable design engineers to be more productive when handling and sharing documentation of designs. Some of the most notable improvements in Vault 2020 were enhancements for handling PDFs. As described in this summary, “[7 enhancements to 2D PDFs in Vault 2020.](#)” many companies can benefit from using the Inventor browser in Vault when using PDFs of 2D design files for internal communication among departments.

Have you run into “incorrect licensing” errors after installing the client during an upgrade? In “[Vault Professional 2018 – Incorrect Licensing.](#)” Dave Morse explains what happened while working on installing his 2018 Autodesk products when he encountered an issue with the licensing mode of Vault.

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STRATEGIC PARTNERSHIPS

Take advantage of manufacturing diagnostics with the Applied Software, Graitec Group customized [Manufacturing Diagnostics](#). You can explore your company's design automation, manufacturing environment, feasibility, and training needs.

With an Applied Software, Graitec Group [Fusion 360](#) Feasibility Study, your team will learn remotely from Inventor experts how to incorporate Autodesk Fusion 360 into your workflow based on your current manufacturing environment.

TRAINING

Take a strategic approach to part and assembly design. [Inventor advanced assembly modeling training](#) topics include: advanced sweeps, lofts, multi-body design, surface modeling, coils, generative shape design, and freedom modeling.

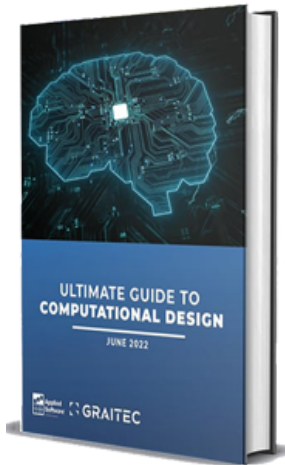
Evaluate your workstation and Inventor environment. The experts at Applied will conduct a diagnostic and environmental [analysis](#) and compile a report on your team's readiness for design and engineering in Inventor.

HOW TO BUY INVENTOR

Inventor can be purchased as a [standalone subscription](#). It is also available as one of the entitlements in the [Autodesk Product Design & Manufacturing Collection](#) of industry-specific software. [Contact](#) Applied Software, Graitec Group to request a quick discovery call and talk to an Applied expert about your company's specific product design requirements.

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