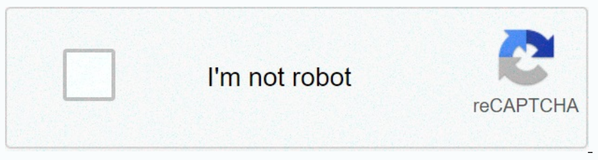

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Fusion 360 tutorial

Autodesk Fusion 360 is a versatile CAD package with all the features needed to develop products from the conceptual phase to manufacturing through design verification and manufacturing on traditional and digital manufacturing tools such as 3D printing. In this article, you'll find a quick overview of Fusion 360's interface and features for 3D printing as well as detailed tips to help you prepare parts for 3D printing. Fusion 360 is a cloud-based CAD platform that is an affordable and highly capable alternative to other major players in the industry. It's easy to use and has all the common features you can expect from popular CAD packages. Fusion 360 was built from the ground up to be a global product development solution and aims to offer a simple workflow of conceptual design throughout manufacturing. Fusion 360 has a very large knowledge base that fully covers all the features of the software, these tutorials can be accessed via Fusion 360 as well as through the Autodesk website. There is also an official YouTube channel with many hours of free tutorials. The software receives frequent upgrades and new features arrive every few months. Fusion 360 is ideal for high-revenue companies as well as start-ups looking for a professional tool to bring them to market. Fusion 360 can perform resource-intensive operations on the cloud, including rendering, simulation, shape optimization and generative design. This means that the work can continue while all heavy lifting is done on the cloud. There are various license packages available, these are listed below: Free Trial: Autodesk offers a free 30-day trial when signing up. Education: Like most other CAD packages, Fusion 360 comes with an educational license to students, educators and educational institutions. Start-up: A free license is available for start-ups, enthusiasts and hobbyists. To use this license, the user must run a business with a turnover of less than $100,000 per year. The start-up license does not include any of the most advanced features such as generative design. Standard: There were two versions of the paid license, namely a standard and ultimate, but these were merged into one version that contains all the features that were in the ultimate version. Fees are structured as a subscription model. In Fusion 360, you can switch between six different work. Each workspace has its own set of tools and functions: Design: To draw 3D models and surfaces using sketches, extrusions, turns, and many other standard CAD tools. Render: Create photorealistic renderings of components and products. Animation: Host assemblies to see if they work as expected or to display features to potential customers. Simulation: Computer-aided engineering to perform various stress analyses on designs to ensure they are operating conditions. Manufacturing: Computer-assisted manufacturing (CAM) to help make the part on various digital manufacturing tools, such as CNC plants, CNC laths, laser cutters and water jet cutters. Drawing: Create drawings for drawing workshops for manufacturing in a traditional manual workshop or to accompany the G code for CNC machined parts. The Fusion 360 workspace is divided into seven main sections: toolbar, app bar, navigation, timeline, browser, view cube and tagging menu. Each of them is described in more detail below. The toolbar contains all the tools and features available in a workspace. In the case of the design workspace, these tools help create and modify 3D models, surfaces, sheet metal and assemblies. The app bar allows the user to open existing projects, create new ones, manually record a project, and access the data panel. The data panel is a space where drawings can be recorded and organized in an easy-to-navigate format. The data panel allows you to create project folders and a place to find examples of parts and tutorials. The navigation bar contains all the tools for rotating, translating and modifying a model's visual style. There are also options to break the canvas into subsections with each indicating a different view of the model. The calendar shows a history of all the operations done to create the piece. This includes all features, patterns, hardware changes and sketches to name a few. This is a unique feature that allows you to see the full history of their game without having to navigate through the browser shaft. Any feature can be changed with a right click in the timeline. The timeline can also be used to find this specific feature in the browser shaft. The browser contains all the components, features, body, sketches and construction geometry of a design. The browser takes the form of a tree-shaped structure that should be familiar from common CAD packages. The view cube allows you to manipulate the model in a more structured way. By clicking either on the corners, edges or faces of the cube, the model will be re-oriented inside the canvas. This make it easy to switch from one standard view to another. If the user clicks on the arrow at the bottom right of the view cube, a drop-down menu appears that provides more options to control the view. The tagging menu is a pop-up menu specific to the which contains commonly used features, it can be accessed by clicking directly on the model or canvas. The features that appear in the menu are determined by what is clicked and what workspace is currently active. This menu increases modeling speed and convenience. The shape characteristic, indicated by a purple cube, allows the sculpture of complex organic forms. It opens up a new workspace that has a wide range of features for sculpting complex shapes. This feature is is to create organic and artistic models for 3D printing. Surface tools allow the user to repair models for 3D printing. Surfaces can be sewn closed, extruded, pulled and pushed to reshape the room. Surface tools also help create a waterproof pattern that has no openings in the shell. This feature is useful for simulating how your 3D printing will behave under loads and dependency speeds over time. For example, break adjustment joints can be modeled to show which loads are experienced by the clip while it is forced closed, providing a good idea of where the weak points are so that the design can be optimized. Some 3D printing technologies such as merged deposit modeling (MDF) create parts with non-linear hardware properties that can only be simulated if the FEA package has a non-linear study type. Fusion 360 has a very capable type of non-linear study that can accurately predict stress on a component provided the correct hardware data is loaded. Learn more about isotropy in 3D printing. Designing for 3D printing is not too complex a task, however, there are some general guidelines to ensure a perfect impression every time. First, consider the 3D printing technology that will be used to print the part. This will determine the type of design constraints, achievable accuracy levels, and support structure requirements Learn more about the three most established technologies for 3D printing plastics today — stereolithography (SLA), merged deposit modeling (FDM) and laser selective sintering (SLS) — in our comprehensive guide. The thickness of the wall is critical because rooms with walls that are too thin will be brittle and could break during 3D printing or post-processing. The recommended minimum thickness of the walls varies depending on the printing technology. Download the Formlabs design guide for 3D stereolithography printers. Overhangs are easy to print on 3D SLS printers because the material is supported by undedified powder. Other technologies such as SLA or FDM may require support structures for overhanging entities. Depending on the technology, the printing of large solid or flat components can cause deformation due to the build-up of heat in the room. Add design features like ribs to make the design structurally stronger and help minimize deformation. Read our guide for ten insights to help you optimize your workflow Rapid 3D printing prototyping to be as cost effective and economical as possible. Clicking on the Make icon in the design workspace opens the 3D printing menu to make a number of changes to optimize the model for printing, and then send the model to a 3D printing utility. The menu is divided into a number of options listed below: this option allows the user to select the model for 3D printing. This checkbox displays the mesh on the model which is useful if the user wants to see what effect the changes in the settings have on the model. This shows the number of that make up the model. Higher refinement will increase this number. This option allows you to select one of three predefined refinement settings: low, medium and high. This determines the total number of triangles used in the model. There is also a custom option that allows the user to further refine the mesh based on specific parameters: Fusion 360 allows you to send the model to a range of 3D printing utilities such as Meshmixer or Formlabs PreForm for 3D printing. Use PreForm software to prepare parts for 3D printing on Formlabs SLA 3D printers. If the 3D printing utility is not selected, Fusion will export the model as an STL file based on the selected refinement options. This STL file can then be loaded into any 3D printer slicer software. Fusion 360 is a versatile CAD set with a range of features that makes it easy to connect digital workflows and move from design to manufacturing. Looking for the right tool to turn your design into reality? 3D printers allow engineers and product designers to quickly prototype in-house, saving time and cost at every stage of product development. Learn more about 3D printing and see first-hand quality by requesting a free sample portion printed on the Formlabs SLA 3D printer. Learn more about 3D printing