CAD Design Resources

Overview

Open Assistive Technology (OpenAT) devices often use computer-aided design (CAD) tools to create the design and generate the necessary files for fabrication. For example, a CAD tool is used to create a 3D model, then create an STL file so the design can be 3D printed.

The goal of this CAD Design Resource is to point makers to useful resources to help them build up their skills using CAD. We direct people to external resources and best practices that can help them learn how to edit designs and create their own devices.

While we have included links to resources we have found useful in the past, we cannot guarantee that external links will remain active.

If you have any suggestions to improve this resource, you can add an Issue to the GitHub page for our resources.

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Getting Started with CAD

Computer Aided Design (CAD) allows users to create digital models of their designs. These can be used by themselves or other designers to create 3D models for manufacturing with a variety of methods, including 3D printing. To start CAD modelling, you need to choose a CAD program.

There are many CAD programs available, whether for free or for different paid versions. If you are designing something using a CAD program that has multiple license types, make sure you are following the rules associated with that license. Some versions restrict the software usage based on the license (ex: student licenses cannot be used to design devices to be sold commercially).

Common CAD Programs

The listed programs are not affiliated with Neil Squire or the Makers Making Change program, and are included here only for informational purposes. We do not endorse any of these products over another.

Autodesk TinkerCAD

TinkerCAD is a free online software for creating CAD models. It requires an account. It uses a graphical user interface (GUI) where you combine 3D shapes to create your design. TinkerCAD is a good starting point for those that are new to CAD.

Autodesk Fusion

Fusion is a cloud-based commercial CAD software developed by Autodesk. It is a paid subscription service with different payment tiers based on number of licenses and use cases. Autodesk products are often used in commercial engineering firms as they are powerful CAD tools. A hobbyist license is available at no cost.

Fusion works through a GUI, where you build up models from 2D sketches into 3D pieces.

Some institutions, such as schools or libraries, may have licenses that can be used by people who attend the school or access the library.

SolidWorks

SolidWorks is a commercial CAD software developed by

OnShape

OpenSCAD

OpenSCAD is an open source CAD software. OpenSCAD works by coding your design rather than using a GUI to create it. For example, you write lines to define a cube at a certain location instead of dragging and dropping a cube onto your screen. The coding interface makes it very popular with users with a programming background.

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OpenSCAD provides a way to set parameters that can be adjusted by the user, without having to modify the more complicated code for the design. This can provide a way to make it easy for a user to create a customized model based on numbers they input.

For example, the [Customizable- 3D Printable Keyguard for Grid-based- Free-form- and Hybrid AAC Apps on Tablets designed by Volkswitch](#) uses an OpenSCAD script to create a customized keyguard based on a number of different options including the type of tablet, app,

CAD Best Practices

Regardless of the software you choose, there are some common best practices for designing CAD models. Some of these best practices are specific to designing for 3D printing. [Protolabs](#) offers some great general tips for designing for 3D printing.

Parametric Modelling

[Parametric modelling](#) is when you use variables and mathematical expressions to define shapes and distances, rather than setting them directly with numbers. Using parametric modelling makes it easier to update designs as you can change the value of the variable and update the model, rather than finding and updating that value everywhere it was used. While it is a useful tool, parametric modelling does make models more complex, increases file size, and is often not as fast as direct modelling.

We recommend using parametric modelling as it makes it easier to improve designs after feedback is received, and it makes it easier to customize designs for other users.

Filleting and Chamfering Edges

Adding fillets (rounding an edge) and chamfers (creating a sloping edge instead of a 90 degree corner) helps to strengthen designs, make them nicer to handle, and improve 3D print quality. [Filleting and chamfering](#) decrease stress concentrations in sharp edges, increasing strength of parts. They also are nicer for a user to handle because edges are less sharp, and they reduce the likelihood of elephant's foot or print bed adhesion issues when 3D printing.

Overhangs

As 3D printed parts are built up, new filament is deposited onto existing layers. If you design an overhang, some or all of the new layer will be printing over empty space. If possible, eliminate overhangs in your design, but try to keep them above 45 degrees (from horizontal) if you need them to avoid the need of support material.

Sharing Designs

Most CAD programs have their own file extensions for CAD models, and these extensions often are not compatible with other software. To share designs across CAD programs, we recommend converting the files to .STEP or .STP format, as this is a universal file format for CAD.

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Finding Help

There are many tutorials available for different CAD software on YouTube and each software's online community.

Printlabs offers online education for 3D CAD and 3D printing.

If you are having trouble with building a Makers Making Change device (customizing the design), you can use the Makers Making Change Community Forum and the Assembly Help category to post your questions/issues to the MMC community.