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## FreeCAD How To

**freecad: your own 3d parametric modeler** - freecad is designed to fit a wide range of uses including product design, mechanical engineering and architecture. whether you are a hobbyist, a programmer, an experienced cad user, a student or a teacher, you will feel right at home with freecad. **basic part design tutorial - rutgers university** - covered. it is only intended to introduce the new user to the workbench. with that being said, what this tutorial will do is take the new freecad user through the steps needed to successfully model the part shown in the image below. before you begin before you begin this tutorial, you will want to adjust a few settings in freecad preferences. **getting started with freecad - folk.uio** - getting started with freecad j. rabault 4th september 2017 1 a few words about freecad  
freecad is a parametric 3d modeler, free and open-source (under the [GPLv2](#) license) that **basically we want to take a hand drawn image and convert ...** - freecad, for right now save the file as an svg by going to file->save as... and saving the file as "appleg". again, if you wanted to, now is the time to scale your image to a specific size before importing it into freecad. we'll leave it at it's default size for this tutorial. now it's time to close inkscape and fire up freecad. **in ones head while learning new features. choose one of ...** - freecad is a very big and growing package, and it is a challenge to keep everything learned in ones head while learning new features. the following notes apply to the windows operating system, but they should be similar on a mac or linux. open freecad either from the task bar or the desk top icon. **freecad - wiki.opensourceecology** - freecad will be a general purpose 3d cad modeller. the development will be completely as open source. as a modern 3d cax modellers it will have a 2d component to extract design drawings from the 3d model, but 2d is not the focus, neither are animation and organic shapes (maya, 3d studiomax,

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