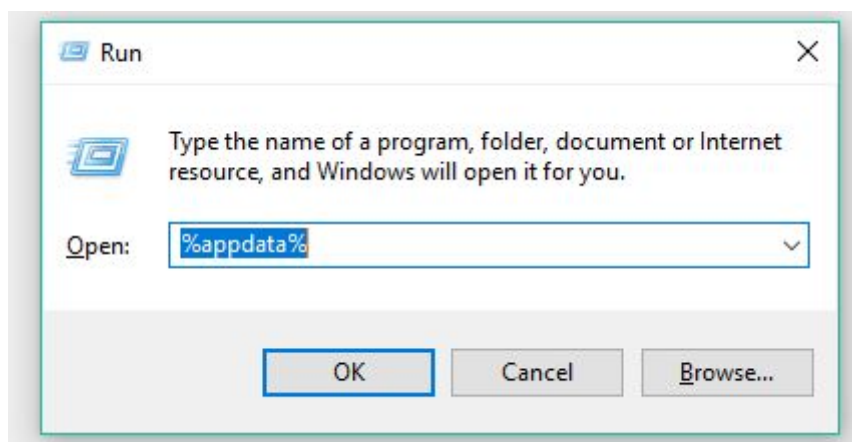


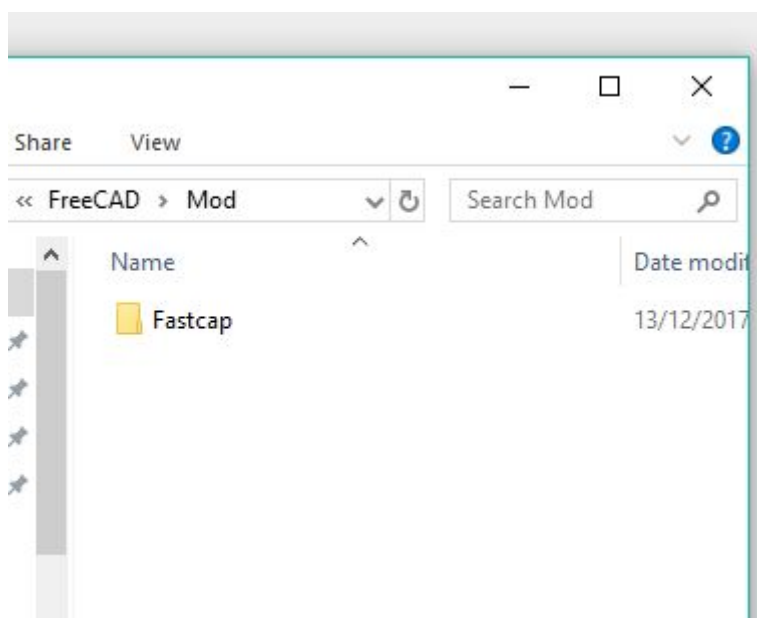
Fastcap Workbench User Guide

Installation

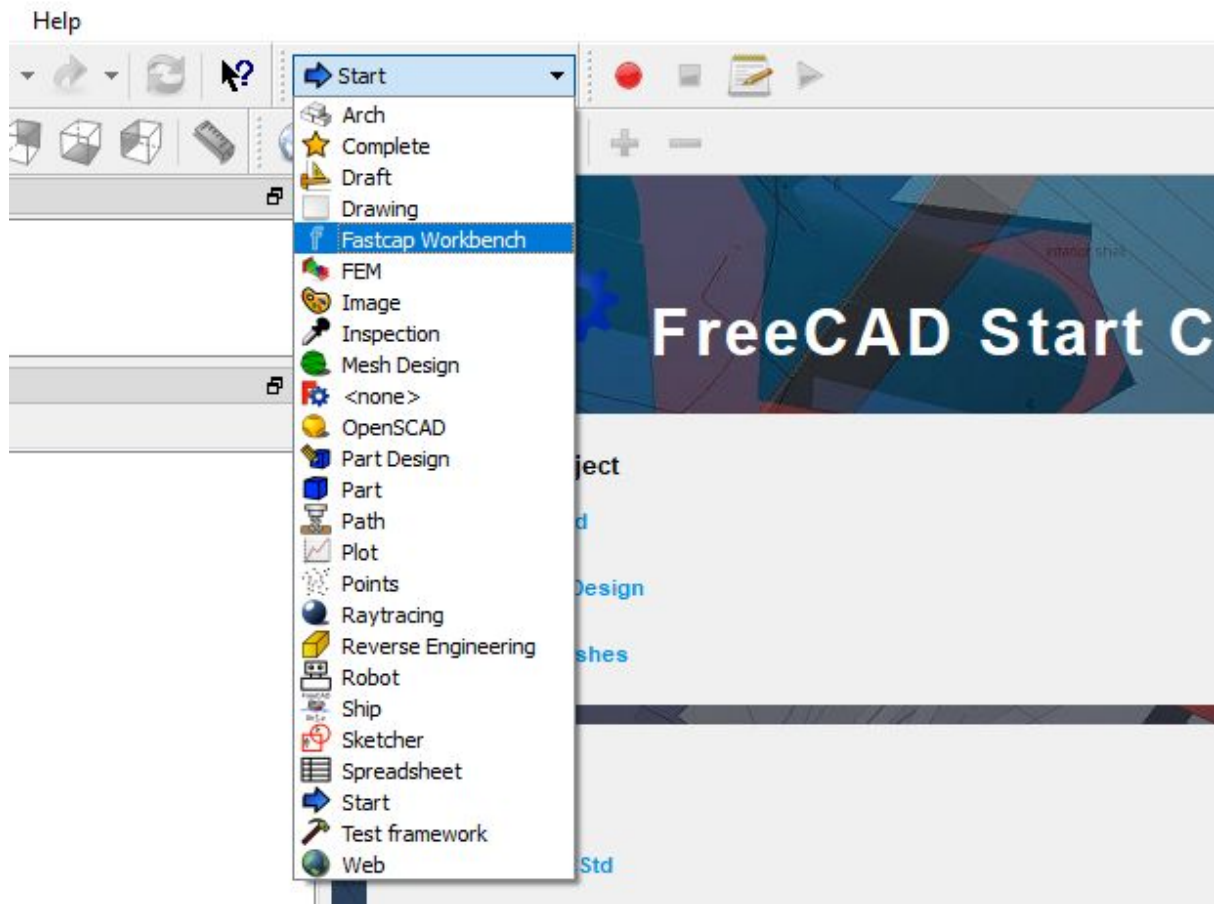
The Fastcap Workbench is an add-on for the open source CAD program, FreeCAD. To install the workbench, first install FreeCAD and any required dependencies. Download the folder containing all the source files from the Github repository and navigate to the appdata folder for FreeCAD. On Windows, this location can be accessed by opening up the run command located in the start menu (or by pressing win+r) and entering '%appdata%' and pressing run.



If there is no folder named 'Mod' located in the FreeCAD folder, create one now. Place the folder containing all the source files inside the Mod folder. Assuming you have no other external workbenches installed, your Mod folder should look like this.



To test, open up a new instance of FreeCAD and see if Fastcap Workbench is an available option in the workbench drop down list here.



If you are able to switch to this workbench without any problems, the installation has been successful.

FreeCAD is the only program required to work with the workbench, however in the case of importing fastcap models as meshes, or working with meshes in general, a dedicated mesh editing tool such as Meshlab is useful for functions like decimation.

Commands

The workbench currently contains 4 tools, all located in the toolbar. The tools will now be described in order of their appearance from left to right on the toolbar.





Import Mesh

The import mesh tool imports a fastcap mesh file (.qui) as a mesh object compatible with FreeCAD and other CAD programs. All quadrilateral panels are thus converted into two triangular panels instead. The function takes any number of qui files and creates separate mesh objects for each file. Depending on the fidelity of the Fastcap file the meshes generated may be quite complex and contain tens of thousands of edges, which will cause problems when attempting to convert to a part. Solutions to this are discussed in a later section.



Export Mesh as Conductor Interface



Export Mesh as Dielectric Interface

The export mesh tools exports a FreeCAD mesh object as a Fastcap compatible qui and lst file. These files will be generated in the same folder where the currently open FCStd file is located. If no lst file exists in the location when a new mesh is exported, a new one will be created. Subsequent exports will append onto this one. If any qui file already exists with the same name of one of the meshes to be written, the existing file will be overwritten.

Upon selecting a mesh and clicking on any of the two exporting functions, a pop up window will open which will request additional information for input into the lst file.

In the case of a conductor interface, the permittivity of the material surrounding the conductor is required. In the case of a dielectric interface, the inner and outer permittivities are required, with the outer permittivity being the side of the mesh containing the normals. To see the calculated normals of the mesh, use the Show Normals functions described below.

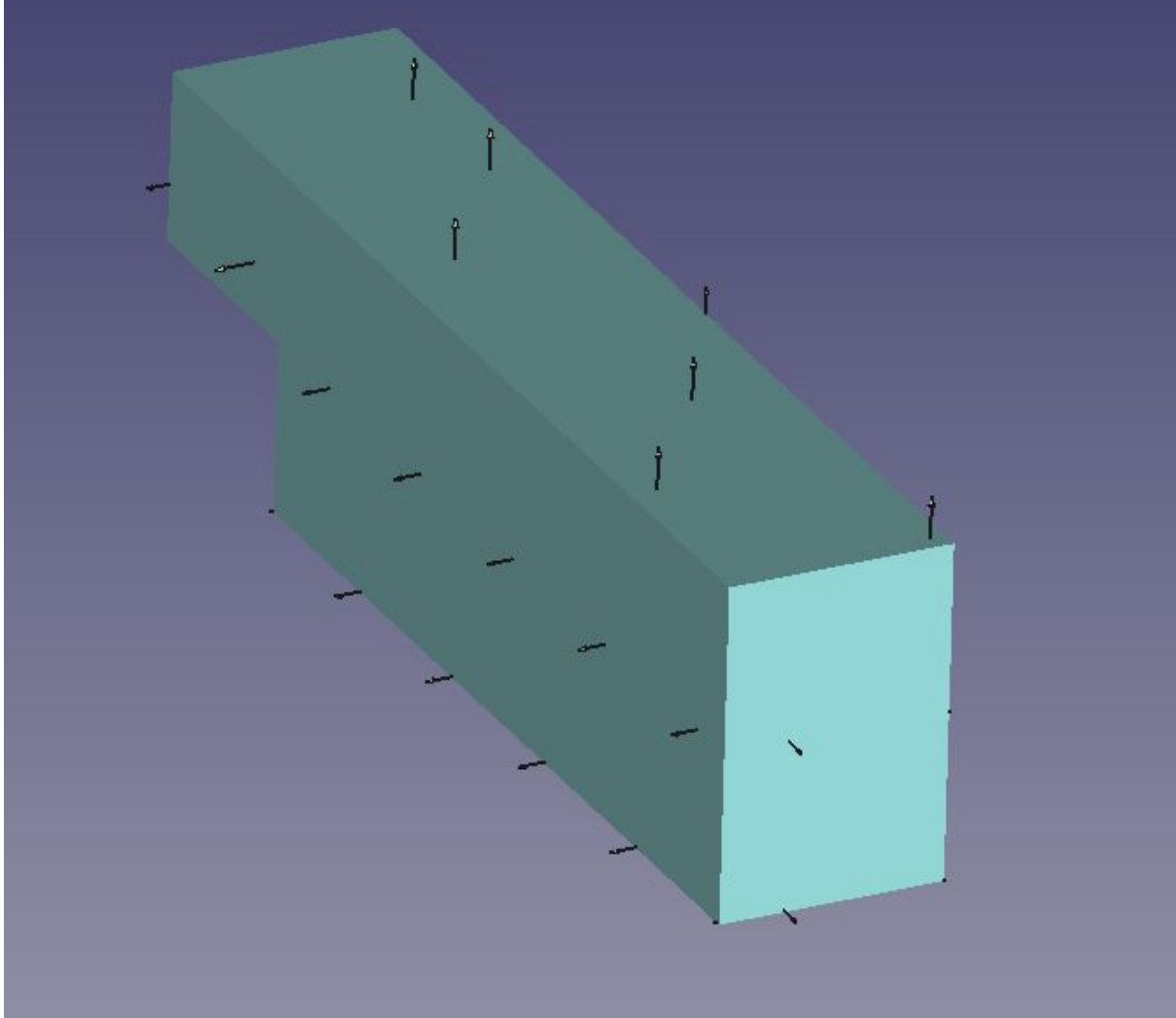
Multiple meshes can be exported at once. Exporting in this way will group the meshes together as one conductor group in the case of exporting as a conductor interface, and make no difference if exporting as a dielectric interface. If a group is being exported, an additional window will open requesting the group name for the conductor interfaces. This group name will be the name shown at the output of the Fastcap simulations and should be named accordingly. If multiple meshes are exported, windows requesting permittivity will open up one after another for each mesh. The title bar of each window will include the name of the mesh currently being exported to help identify what permittivities need to be inputted.

In the case of an error, there is currently no way inside the program to edit the output files. Functionality may be added in a later update, but if a mistake is made, you must edit the relevant qui and lst files yourself, or start over from scratch by deleting the lst file.



Show Normals

The Show Normals tool selects randomly the normals of up to 40 faces across the entire mesh and displays them. Use this function to identify which side the export as dielectric interface will treat as the outer side. See the below picture for clarification.

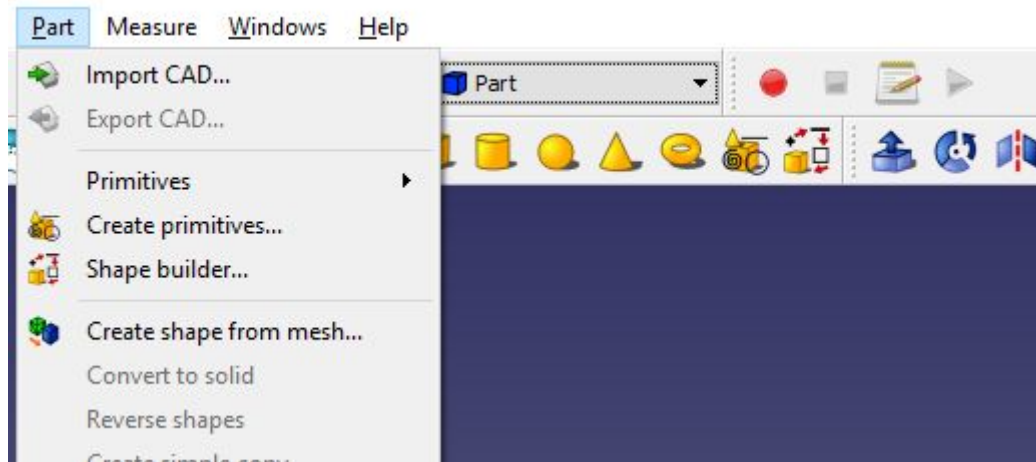


Arrows pointing out of side of the mesh which will be treated as the 'outside'.

Editing and Creating Meshes

Despite meshes being the only format available to be exported as a Fastcap compatible file, FreeCAD offers far more useful object types for defining shapes.

An imported mesh can be converted to a shape by the Create shape from mesh available in the part workbench.



The shape can then be converted to a solid via Convert to solid, and optionally refined with Refine shape as necessary.

As a solid, the object can be used in typical part operations available in FreeCAD to get the required shape.

The required shape can then be converted back to a mesh via the Create mesh from shape function available in the Mesh workbench. The fidelity of the meshing chosen now will directly translate into the fidelity of the panels in the Fastcap model.

One method to generate a good result is through the Mefisto Mesher. The maximum edge length can be changed to specify the fidelity of the meshing. In the case that the same fidelity is not required across all faces of the object, the shape can be split into separate faces (through the Downgrade tool available in the Draft workbench) which can be individually converted to meshes and joined as a single Fastcap group (if they are conductors).

Meshes can also be exported via the Mesh workbench in a format compatible with most programs. If AutoCAD functionality is preferable, the mesh can be imported into AutoCAD, converted and/or edited, then exported again back into FreeCAD to be converted into a Fastcap file.

A common problem with imported meshes from Fastcap files is that they are too complex for the shape conversion to complete. A workaround is to export the mesh into Meshlab which offers a decimation function to reduce the complexity of the mesh. Simple shapes, such as ones with only a few actual faces, can be decreased in complexity by a magnitude of 10 or more without noticeable deformation. The resulting mesh can then be imported into FreeCAD and converted into a shape in much less time.

Another option, if the shape is incredibly simple, is to simply recreate the object from scratch using the mesh as guideline.