

## Converting .stp to .dxf files

For panel layout drawings it is necessary to have 2D CAD files. Here is how to convert a 3D .stp CAD file to a 2D .dxf file using FreeCAD, an open-source 3D CAD software. This will allow you to create 2-dimension views of the the 3D CAD model in dxf form to be used in 2D CAD packages like AutoCAD.

FreeCAD can be found here: <https://www.freecadweb.org/> - For this instruction set, version 0.19 is used.

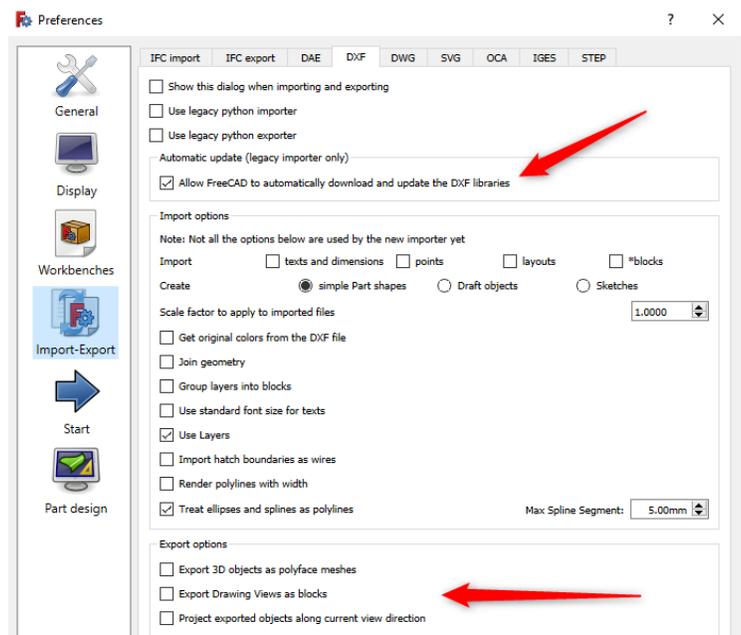
The Rexroth Media Directory can be found here:

<https://www.boschrexroth.com/en/xc/myrexroth/media-directory>

3D .stp CAD files for most products can be found and downloaded here.

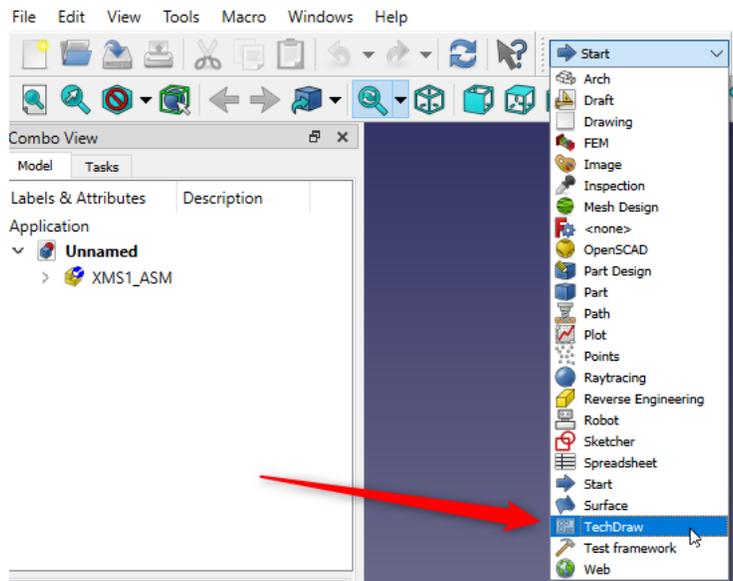
### Prepare FreeCAD

Select "Edit" from the toolbar, then "Preferences" from the drop down. In the "DXF" tab, check the box for "Allow FreeCAD to..." and uncheck all the boxes for the Export Options at the bottom:

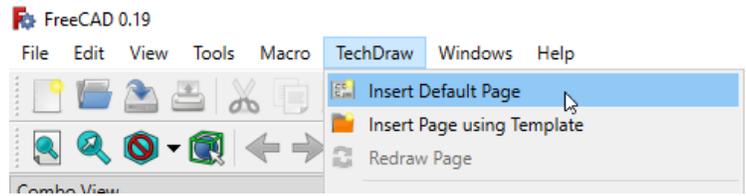


### Create the DXF file

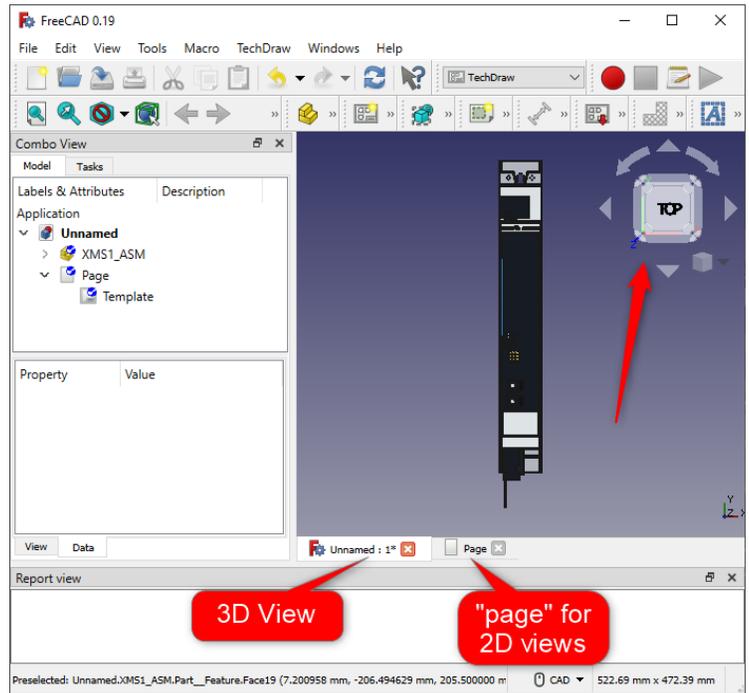
Next, open the .stp file in FreeCAD and once the drawing is open, Select TechDraw from the pull-down menu on the tool bar:



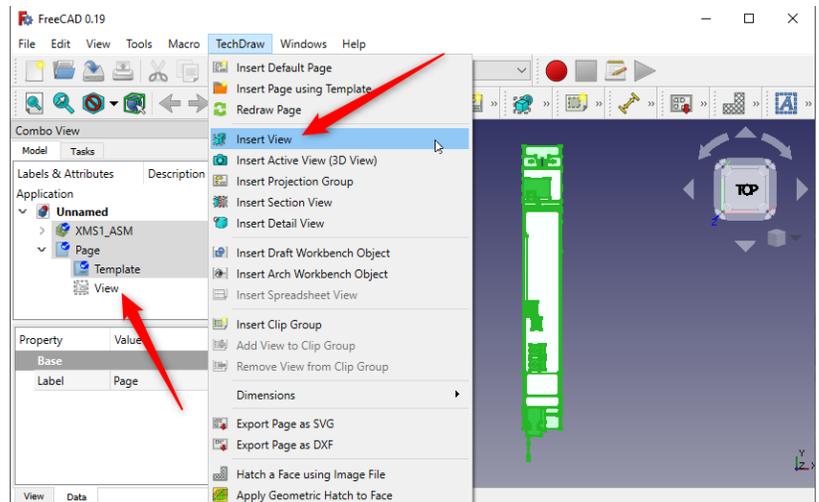
Select the TechDraw menu on the toolbar and select "Insert Default Page". This will insert a default title page that will not be saved to the .dxf file that will be created later.



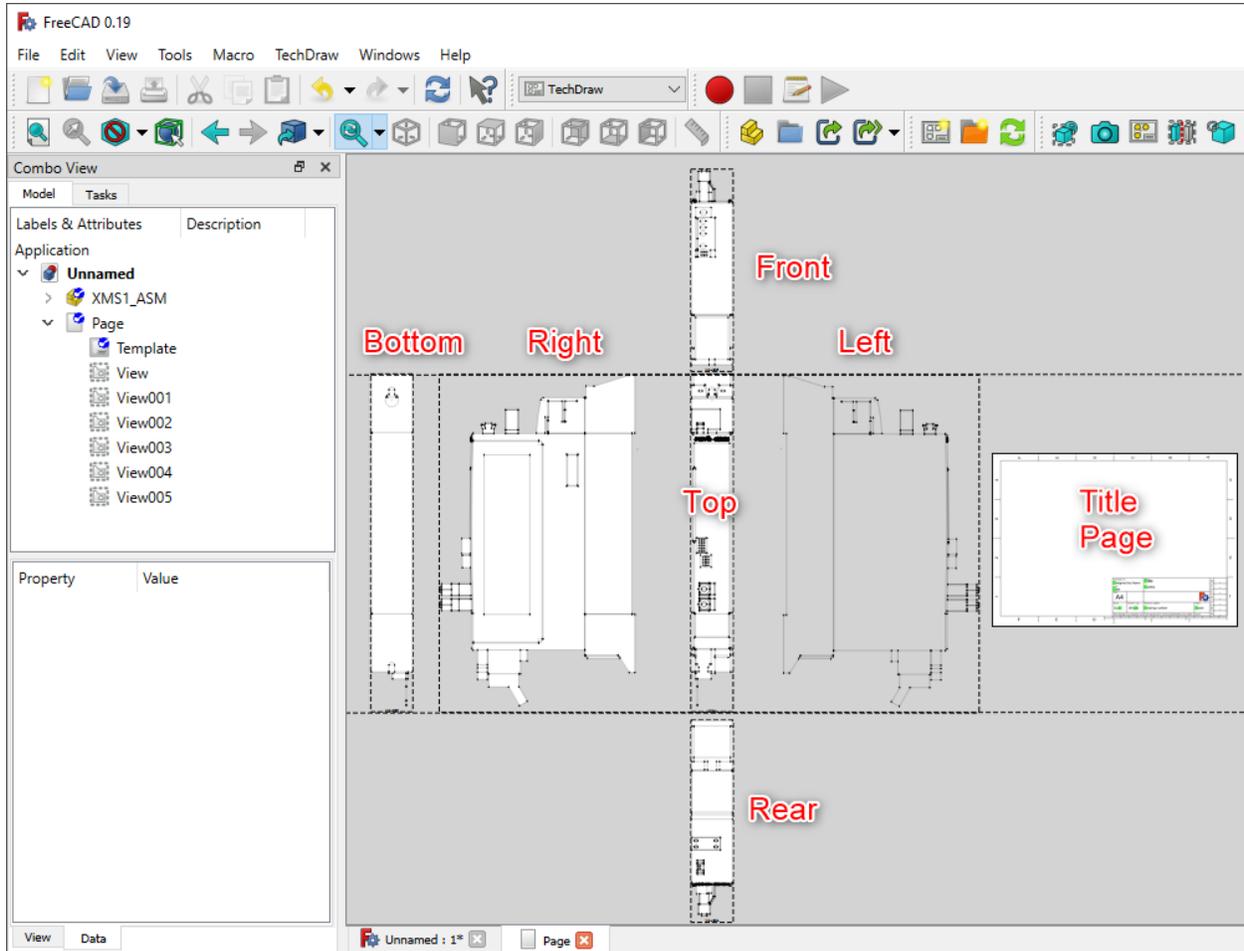
Return to the 3D view and use the orientation tool in the upper right-hand corner to select a view to create. The goal is to create 6 views to put in one dxf file: front, back, left, right, top, bottom. Each view must be created separately using the process below and then the entire page, including all 6 views can be exported to .dxf.



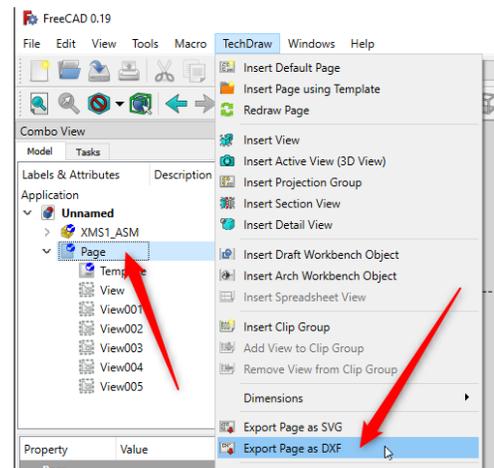
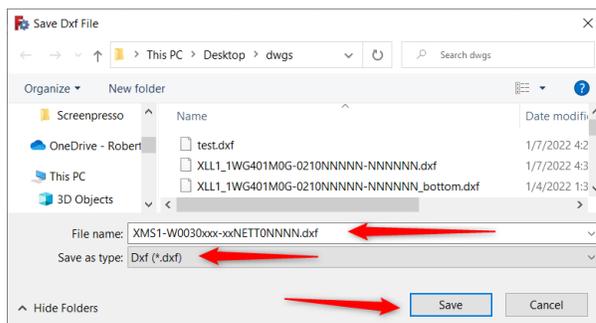
To create one of the six views, orient the model in the desired view, and press "ctrl-A" to select all. In the TechDraw menu, select "Insert View". This will create a "View" under the "Page" created earlier.



Change the orientation to another of the six views, press “ctrl-A” to select all and select “Insert View” from the TechDraw menu. Repeat this process until all six views are created. Each new view is put in the center of the page, overlapping the first view. After each view is “inserted” return to the page view and move the new view to the side where it will be visible. The end product should up looking like this:



Next, select the page in the Application tree and select “Export page as DXF”. Select a file name and make sure it is a .dxf file type.



After this, the Project can be saved as a .FCStd file that will preserve the page and view configurations that were created.

## Open the .dxf in AutoCAD

Often there are conversion errors that must be corrected before the file will open in AutoCAD. For the purposes of this explanation, AutoCAD Electrical 2020 is used.

In some cases, when attempting to open the dxf file, drawing components are corrupted and must be removed. The indication that this has happened is that a prompt "Press ENTER to continue" is presented to the user and when Enter is pressed, the opening of the dxf file crashed. Typically, in the message bar, the offending line of the dxf file will be displayed. In this case open the dxf file with a text editor such as Notepad. Use the "Go to..." feature in the Edit menu of Notepad to go to the line of the error as indicated by AutoCAD. Delete this entire object. Remove any empty lines and save as a new .dxf file. Try to open this new .dxf file and see if any new errors exist. If not the .dxf file will open. If errors still exist, repeat this process again until no errors exist. The final file will look like this:

