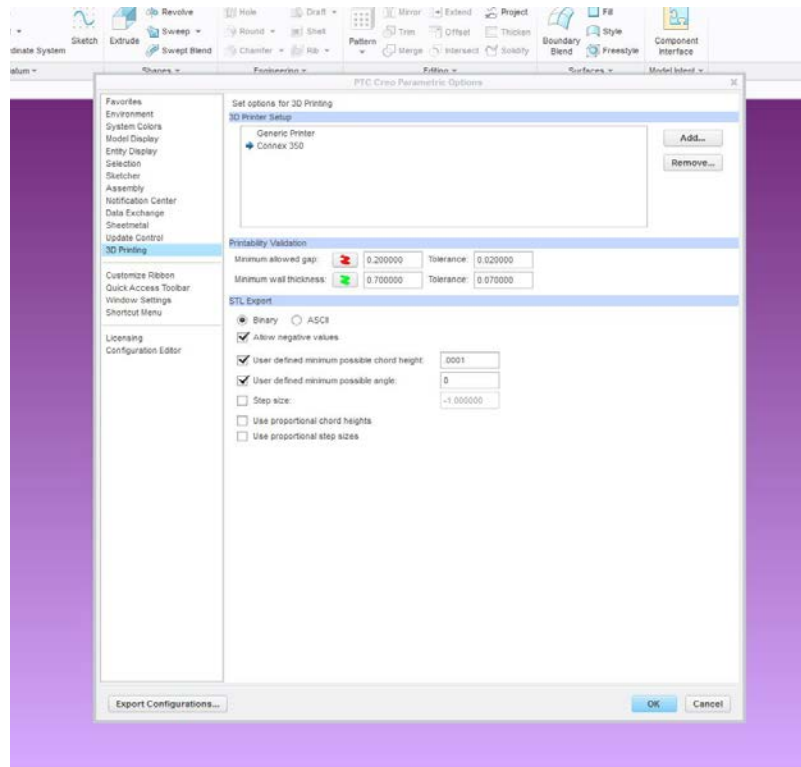# Optimizing 3D models to STL Files in CREO

There is a command in CREO that will allow you to optimize your design for 3D printing. To find the command go to *FILE* and choose *OPTIONS, 3D Printing* and you will see this menu come up:


What this command can do:

- Identify printability problems with your design, such as thin walls or narrow gaps that would wash away during the cleaning process.

- Allows you to position and scale your model for the best efficiency and cost effectiveness.

- View the model showing a cross section to be insure there are no hidden empty pockets.

- Save the CAD data in a *.stl* file (preferably NOT in your project directory).

Procedure:

1. Open your part or assembly in CREO.

2. Refer to Guide #5 – Creating STL files on the D&D guidelines page at
   https://www1.aps.anl.gov/Design-Drafting/Design-and-Drafting-Procedures

3. Open your *.stl* in CREO and proceed.

4. Click *File > Print > 3D Print* to open the tool. This will display a new Ribbon/Tool Bar.
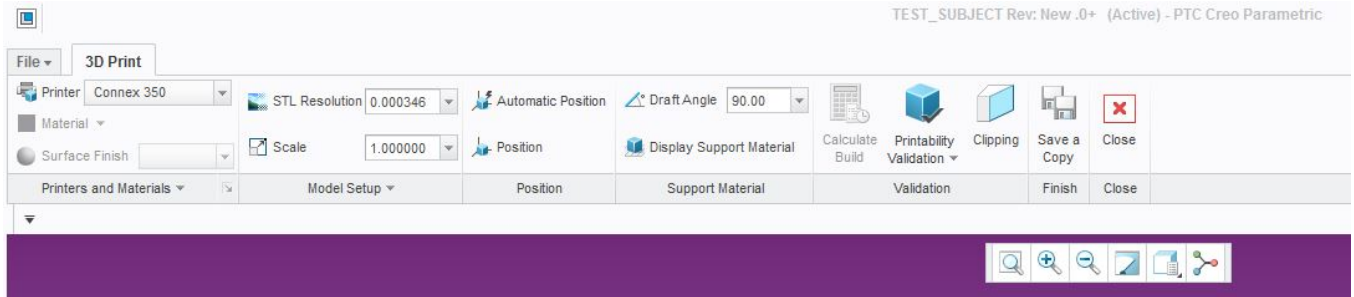
5. Click *Printer* and choose the printer. If Connex 350 and Objet30 Pro are not listed, they may be added by closing the 3D print tool and going back to your part or assembly. Choose **FILE > Options > 3D Printing**; this will bring up the page you see at the beginning of this document. Choose **Add** and enter data in the illustration below:


Now you will have an accurate representation of the build table for your model. Make sure that under **STL Export**, **Binary** is checked and **chord height** and **minimum possible angle** are set to **zero**.

6. Open the .stl you created in CREO and click *File > Print > 3D Print* to open the tool.

7. Your model will be displayed and positioned on the 'tray'.

8. Click *Position* to rotate and position the model on the tray by dragging it about. The *Automatic Position* button will place the model in the best position and orientation.
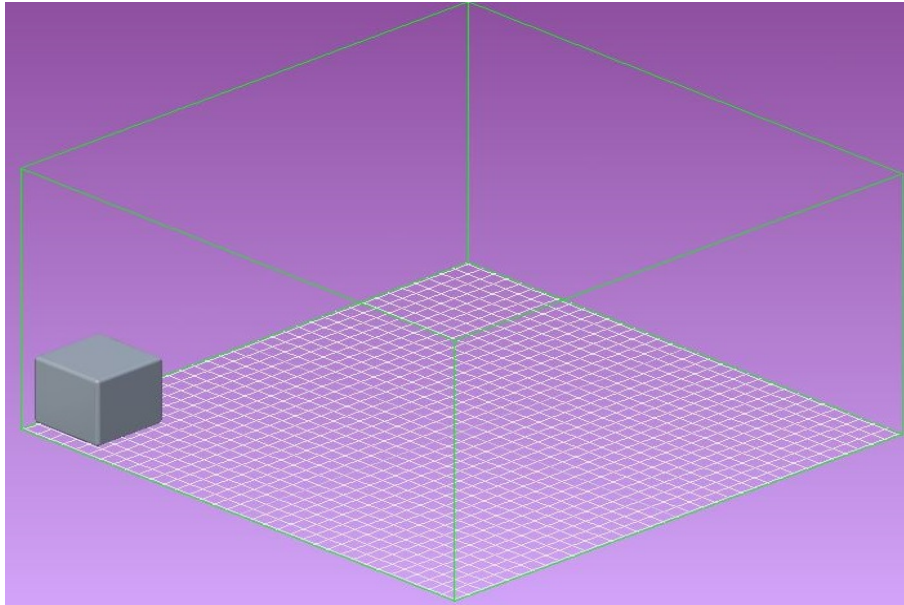
9. Click *Display Support Material* to see a representation of the support material needed for the part. (The Orange color at the base of the cube is support material.)
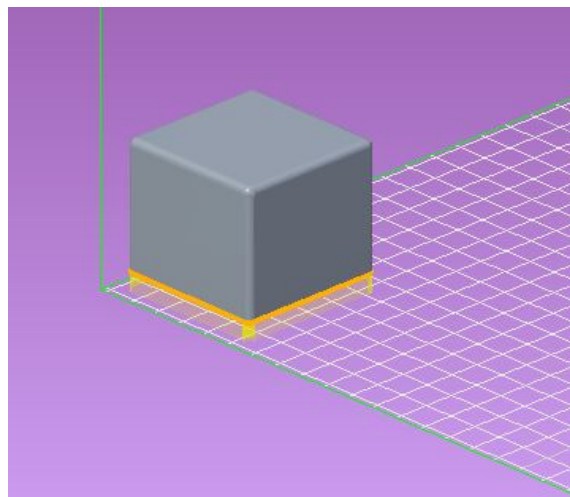
10. Click *Printability Validation* and the drop down allows you to do a complete validation or one to check for *narrow gaps* or to check for *thin walls.* The checks allow you to see where a design may need adjusting.

11. To view the interior of the model and the support material at various points, click the **Clipping** button and use the 3D dragger to display the interior of the model.

12. To save a copy of the model, click **Save a Copy** button. The Export STL dialog box opens, accept the default name or type a new name for the STL and click *File > Save.*

13. Please note that **Calculate Build** is grayed out and is unavailable. The 3D printers are not connected to the general network to keep them secure.

14. Click **Close**.