

## Creating flat parts in Creo that can be imported in sheetcam as DXF files

1. Open Creo
2. Click NEW, in the top left corner
3. Choose part under type, and solid under sub-type.
4. Create the part in Creo
  - a. Under model, sketch the part and click “ok” when finished.
  - b. Under model, extrude the part to the desired sheet thickness and click “ok” when finished.
  - c. Under Tools, click “family table”
    - i. After the table opens, click “insert a new instance at the selected row”
    - ii. A list of instance file names and common names will show up, change both bottom names to (partname)\_cut
    - iii. Click “ok”
5. Save this designed part as a “prt” file. (prt files are exclusive to Creo)
  - a. Click “File”
  - b. Hover over “Save As”
  - c. Click “Save a Copy”
  - d. Choose the file destination, name it, and save it as a “prt” file.

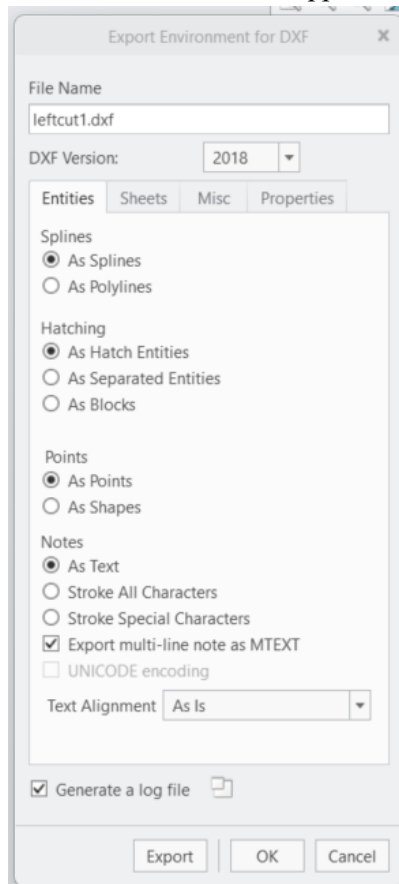
At this point, you will have a 3d model of the part you want after the plasma table cuts it in your Creo window.

6. Click “New” again, and choose “Drawing” as the type, and name the file.
  - a. Click “Empty” under specify template, choose “landscape” under Orientation, and then choose the size.
    - i. For size, there are several options, these options consist of inches and millimeters, as well as width and height of the drawings ‘size’. (From my understanding, the size has more to do with the size of paper it would be printed on, not the actual size of the part.)
  - b. Under Select Instance, highlight “The generic” then click “Open”
7. The drawing layout will be displayed.
  - a. Click “General View”
  - b. Highlight “No Combined State”, click “Ok”
  - c. Click somewhere inside the box and your part will import, and a window will pop up.
    - i. Highlight “View Type”
    - ii. Change the View name if you desire
    - iii. Under View Orientation choose “Views names from the model”
    - iv. Highlight the views from the options below and click “Apply” until you find the view you want.
    - v. Highlight Visible Area and choose “Full View”
    - vi. Highlight scale and make sure you have it set as 1:1.
      1. This could be under default or you may need to change it to custom scale.
    - vii. Skip View States

- viii. Highlight View Display.
  - 1. Under Display Style choose “No Hidden”
  - 2. Under Tangent edges display style choose “None”
- ix. Skip Origin and Alignment unless you care where they are sitting on the ‘paper’.
- x. Click “Ok”

You should now see a ‘wireframe’ view of your part, and the border of the sheet. We need to get rid of this border or sheetcam will recognize it as a cut path.

- 8. Click “Sheet setup” under the Layout tab.
  - a. Uncheck the “show format” box.
- 9. Save the Drawing
  - a. Click “File”
  - b. Hover over “Save As”
  - c. Click “Save a Copy”
  - d. Choose the file, name it, and save it as a “dxf” file.
  - e. The window below will appear, I leave all the default clickable options and click “ok”.



- f.
- 10. Open Sheetcam TNG
  - a. Click “File”, and then choose “import drawing”
  - b. You should now see your Creo designed part in Sheetcam.
- 11. From here, you can use sheetcam to choose your tooling, modify the cut paths, and then save it as a Gcode file.

Disclaimer: This is the way I have come up with for importing Creo designed flat parts into sheetcam. These are not published instructions from Creo or Sheetcam. Parts with bends will need to be laid flat before creating the file in the family table and saving them as prt files.