Lab Assignment #5: Intro to Part Modeling ME 170

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Date Due: One week from Start Day of Lab (upload deadline – midnight before next lab)

You will learn the basic operation of the sketcher in Creo and how to capture "Design Intent" through use of 'constraints'. You will then CAD model a few simple parts.

Part I. Create a Creo template called '170startpart'.

This file will be used throughout the class as the starting template for all new parts.

1) Start Creo, set the working directory, and create a new part.

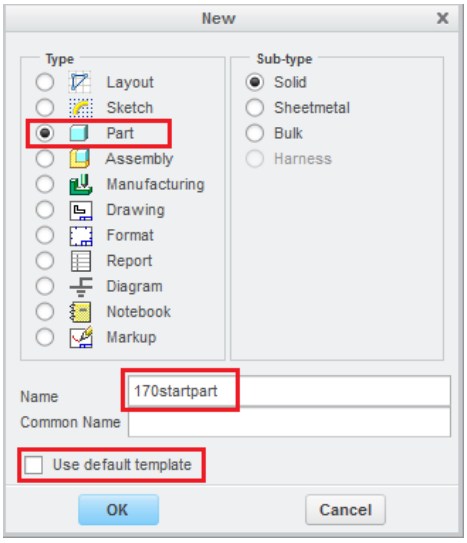
   a. Start Creo Parametric 5.0.

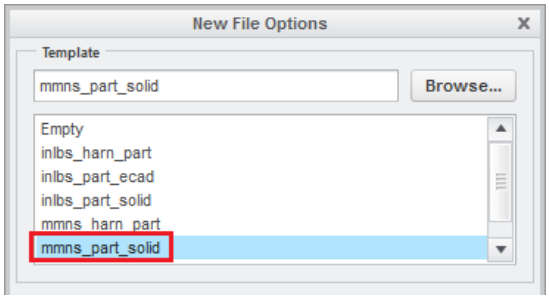
   b. Set the working directory to your "ME170" on the desktop. See Lab 4 for detailed instructions on setting the working directory.

   c. Create a new part by navigating to File → New.

   d. Enter '170startpart' as the part name. Uncheck the "Use Default Template" box. Click OK.

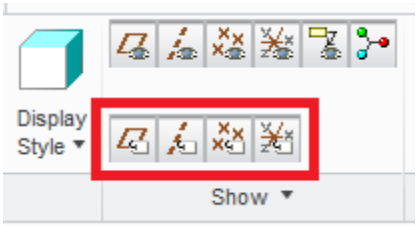
   e. Select 'mmns_part_solid' from the "New File Options" dropdown menu. Note that this template creates a new solid part with the units set to millimeters-Newtons-seconds.





2) Display the datum tags.

   a. In the "Show" section of the "View" ribbon, toggle "on" the datum tags for planes, axes, datum points, and coordinate systems (image below). Each of the four datum tag display buttons should appear depressed.



3) Practice renaming datum planes.

   a. From the “Model Tree” (the list of items on the left side of the screen), double left mouse click on “Right” and rename it “DTM1”. Alternatively, select (left click) the right datum plane in the 3D model view window. Then, right click and hold to show an editing menu. Select “Rename” and type “DTM1”.

   b. Reverse the above operation. Change the name of the datum plane back to "Right".

      Note: Most features in the model tree can be renamed. Is often useful to rename features and planes when creating parts to assist with future editing and modification of part files. "Oil_pan_holes" is a much more useful descriptor than "Hole id 12."

4) Save the file.

   a. Navigate to File → Save. Click OK.

   b. Close this part by navigating to File → Close. Do NOT select File → Exit. The "Exit" button will close Creo, whereas the "Close" option will merely close the part.

   c. If you accidently click "Exit" and close the Creo program, you will need to set your working directory upon restarting the program.

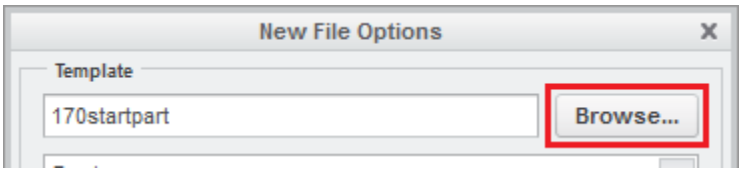
Part II Creating a New Part.

Every part we will be making will follow the same initial steps to setup the file. Follow the directions below for each new part.

1) Create a new part using the 170startpart template.

   a. Navigate to File → New. Enter "<netid>_partname" for the part name. Uncheck the "Use default template" box. Click OK.

   b. Click the "Browse..." button. Select your "170startpart.prt". Click "Open". Click OK.



Part III. Complete Exercise 3 - Tasks 1, 2, 3, 4, 5 & 6 only (pg. 80-88)

In this exercise, you will begin modeling the textbook's small electronic device. You will create some part files (.prt) and submit them for grading. The notes included below should assist you with the exercise.

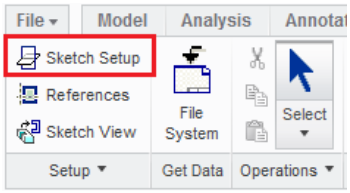
Before Beginning

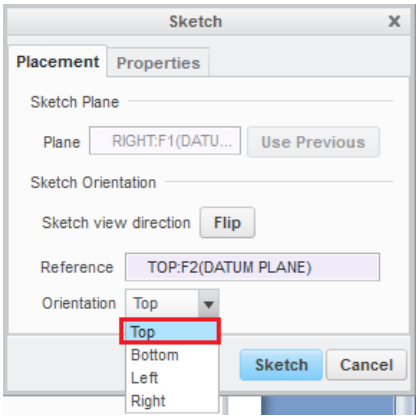
- Ensure that your working directory is set to your "ME170" folder on the desktop.
- Remember to add your NetID to all new part files. The book will ask you to create a part named '4455-001' for example. Be sure to add your NetID to the part file name: "<netid>_4455-001".
- Use the "170startpart.prt" to create all new part files.
- Save often throughout the exercise.
- To add dimensions to a feature use the dimension tool:

For regular linear dimensions, left click on this icon, then left click on each reference line of the sketch followed by middle mouse click to postion the dimension value (usualy half way between the reference or extension lines

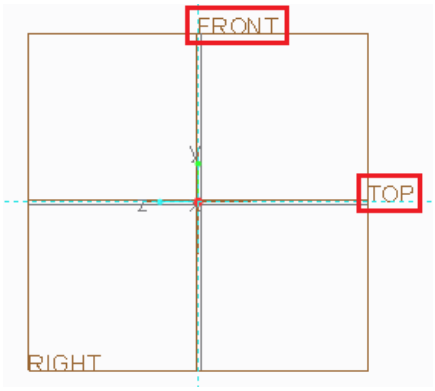
Task 1

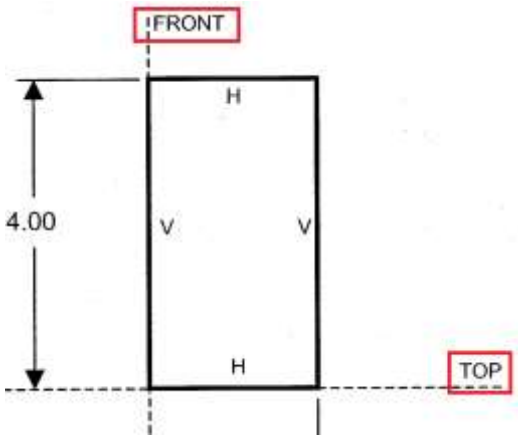
- You do not need to start a new Creo session if Creo is still open.
- Do not add or apply the configuration options as the book instructs.
- Remember to click "Sketch View" to properly align the sketch.
- You will need to re-orient the sketch before beginning. You will need to re-orient most sketches when completing book exercises.
    - Click "Sketch Setup" in the "Setup" section of the "Sketch" ribbon. See image below.
    - Change the reference plane and/or orientation until the plane configuration matches that which is in the book. In this instance, only the orientation needs to be changed to "Top".



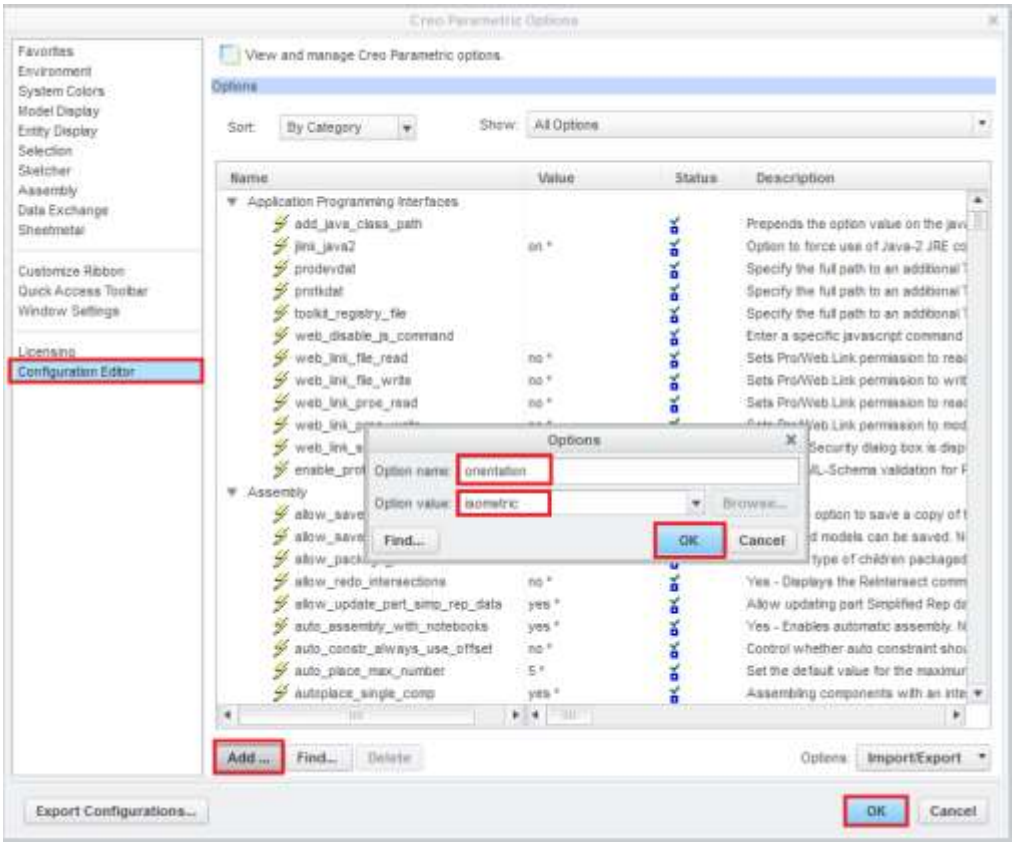


- Check that the orientation of the planes in Creo's sketcher matches what is given in the book. These steps ensure that your parts will assemble correctly at the end of the book exercises.





- The standard orientation on the EWS computers will not match the standard orientation in the book. The book uses an 'isometric' orientation, where as the EWS computers default to 'trimetric'.
  - Navigate to "File" → "Options". Select "Configuration Editor" in the left column (image below). Click "Add...", type "orientation" for the Option Name, and select "isometric" from the drop down menu. Click OK. Click OK. Click "Standard Orientation" under the "View" ribbon, and your sketch, if correct, should match the book's. See the image below.



Task 3

- You may lose some constraints when you apply the fillet tool to the triangle.
    - You may need to re-apply the symmetry constraint to the base of the triangle. Do this by selecting the "Symmetry" constraint on the "Sketch" ribbon and then clicking the triangle's left bottom corner point, the vertical centerline, and then the right bottom corner point.
    - You may also need to re-apply a coincident constraint to the top of the triangle to align it with the vertical reference.

Task 5

- Sketch the second centerline before drawing any lines.
- The workflow that you use to create sketches in tasks 5 and 6 deserves special emphasis since it is the best way to create a sketch. Here, is a simplified version of the workflow:
    - Sketch the shape of the part. You may have to exaggerate the geometry to avoid certain automatic constraints.
    - Apply any constraints. Use the dimension tool to create all of the dimensions. Do not modify the dimension values.

- Use the modify tool to modify the dimension values. Remember to uncheck the "Regenerate" checkbox.

Part IV. Complete Exercise 4 – Tasks 1 , 3 thru 9 only (pg. 105-115)

Before Beginning

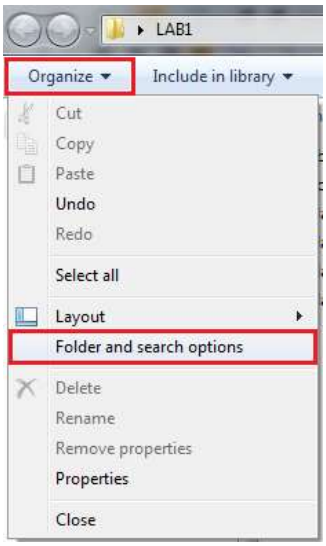
- Ensure that your working directory is set to your "ME170" folder on the desktop.
- Remember to add your NetID to all new part files. The book will ask you to create a part named '4455-001' for example. Be sure to add your NetID to the part file name: "<netid>_4455-001".
- Use the "170startpart.prt" to create all new part files.
- Reorient your sketches using the "Sketch View" button.
- Ensure that the sketch orientation in Creo's sketcher matches the book's sketch orientation. Use "Sketch Setup" to adjust the orientation.
- Save often throughout the exercise.
- Recall that you can set the "orientation" configuration option to "isometric" so that your "Standard Orientation" matches the book's.
- Ensure that all dimensions AND constraints are strong before confirming the sketch.

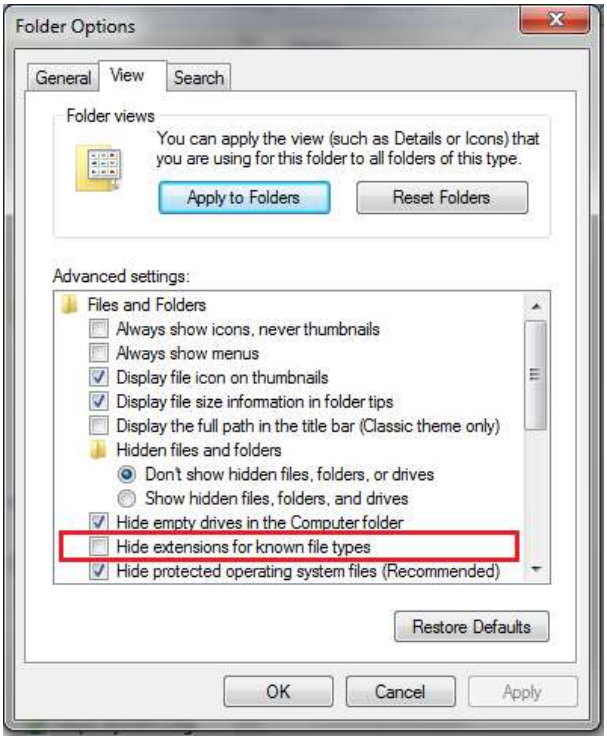
Part V. Submission Requirements

1) Change your file display settings to select the correct part files for submission.

   a. In the windows file browser, NOT the Creo file browser, select "Organize" → "Folder and search options."

   b. In the "Folder Options" window. Navigate to the "View" tab and uncheck "Hide extensions for known file types." Click "OK." Notice the numbers that appear after your part files. Those numbers are the Creo's version designators.





2) Copy and paste the following files to the "Lab5" folder in your "Submissions" folder. Do not cut-paste or remove the files from your ME170 folder; create a copy of them in the "Lab5" directory. Be sure to include the latest version of each part. (Ensure that '#' is the highest numbered file in your ME170 directory.)

   netid_4455-001.prt.# (from Part IV– and text)
   netid_4455-003.prt.# (from Part IV– and text)
   netid_4455-004.prt.# (from Part IV– and text)

   You are required to complete the additional parts (4455-002 and 005) in Exercise 4, but you do not need to submit them at this time. You will work on these parts in later assignments before we ask you to submit them for grading.

3) Once you have double-checked that all the correct files are in your "Lab5" folder, navigate back to the "Submissions" folder. Right-click the "Lab5" directory. In the menu that appears, select "7-Zip" → "Add to "Lab5.zip"".

4) Rename your new zip file to include your NetID: "<netid>_lab5.zip".

5) Submit your "<netid>_lab5.zip" file via the my.mechse website. See Lab 4 for instructions on how to submit files through this website.