MEEG202 LABORATORY SESSION - Dimensions, Tolerances, and GD&T

Please use SolidWorks to create the part shown.

Once you have created the part insert the two views - as shown - on the UD title block and dimension exactly as shown; don’t forget to add any missing hidden lines or centerlines and eliminate any tangency edges. Fill in the title block with the COURSE, DWG TITLE, YOUR NAME, DATE, and SCALE and raise your hand, when finished, for attendance. (Below is a short guide explaining aspects of dimensioning and tolerancing in SolidWorks that you will need to know for this lab and the homework.)

---

**Dimensioning:**

There are several ways in which to dimension a drawing in SolidWorks. The first, and often simplest, method is to insert all the construction dimensions used to create the part in the first place. This can be done by selecting one of the principal views and then going to **Insert>Model Items** in the windows bar and putting in the dimensions for the entire model as you did in an earlier lab. This will insert all dimensions used to create the original part, including extrusion distances, angles of revolution, fillet radii, etc. You can then click and drag to place the dimension text somewhere else, or delete unnecessary ones by simply selecting them and then hitting the delete key on the keyboard. Missing dimensions can then be added using the **smart dimension** tool just as they are in the model space.

An alternative method is to enter all the dimensions manually.
Editing Dimension Text:

It may be necessary to add notes to dimensions or modify the text – such as in thru holes etc. Once a dimension is placed, its text can be edited in the “Dimension Text” box located in the “Dimension” menu of the features menu on the left-hand side of the screen. All text can be added by simply typing it in this box either before or after the <DIM> notation (which links the dimension from the part file). This box also contains a wide variety of common symbols and text orientation options.

Tolerancing:

Tolerancing can be done in much the same way as adding text to a dimension. After selecting a dimension from the paper space, the “Dimension” menu will appear in the features menu on the left-hand side of the screen. The second menu from the top is “Tolerance/Precision” which contains many options to augment dimensions. In this menu you can change the precision (number of decimal places) of dimensions and add one of several types of tolerance options. When adding a tolerance, note that it is necessary to input the minimum and maximum values in the “Tolerance/Precision” window, and not the paper space.

Graphical Dimensioning and Tolerancing:

To add a datum flag to the reference geometry go to Insert>Annotation>Datum Feature Symbol. Then you can place it anywhere on the geometry you wish and change its letter callout in the menu on the left-hand side of the screen.

To add a geometric tolerance to an already placed dimension, first select that dimension, and then go to Insert>Annotation>Geometric Tolerance. This will bring up a text box containing several drop down menus and text fields. Select the appropriate symbol, enter the tolerances, and then the letter of the datum feature you are referencing.

To add a geometric tolerance to an entity that is not dimensioned (such as the parallel tolerance on the layout for this lab) make sure you do not have any selected dimensions and go to the same tool Insert>Annotation>Geometric Tolerance. This will place a free geometric tolerancing box that can then be dragged to the area you wish to place it in. To add a leader go to Insert>Annotation>Multi-Jog Leader and then use your mouse to draw the leader from the geometric tolerance to the desired feature on the part.