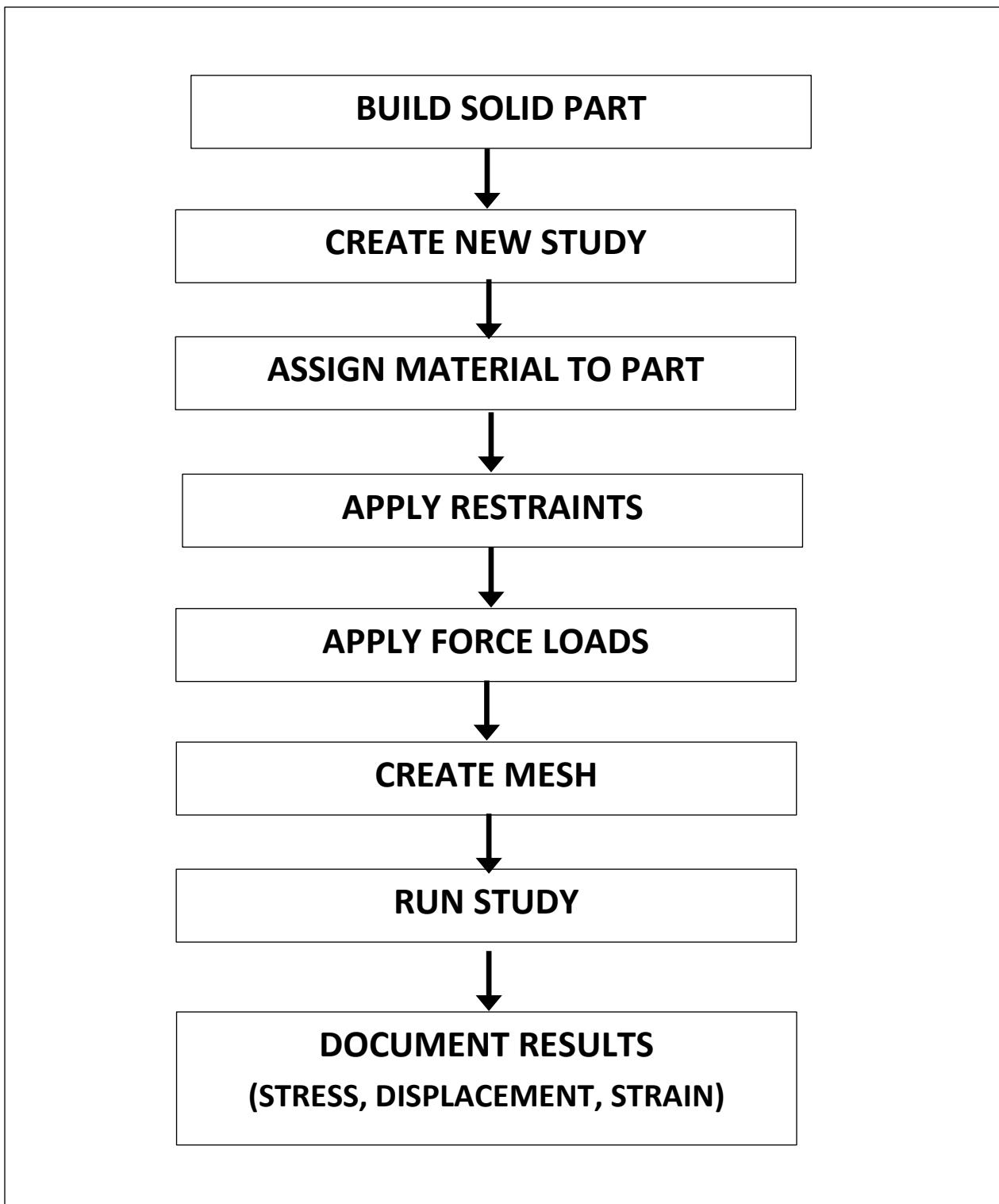


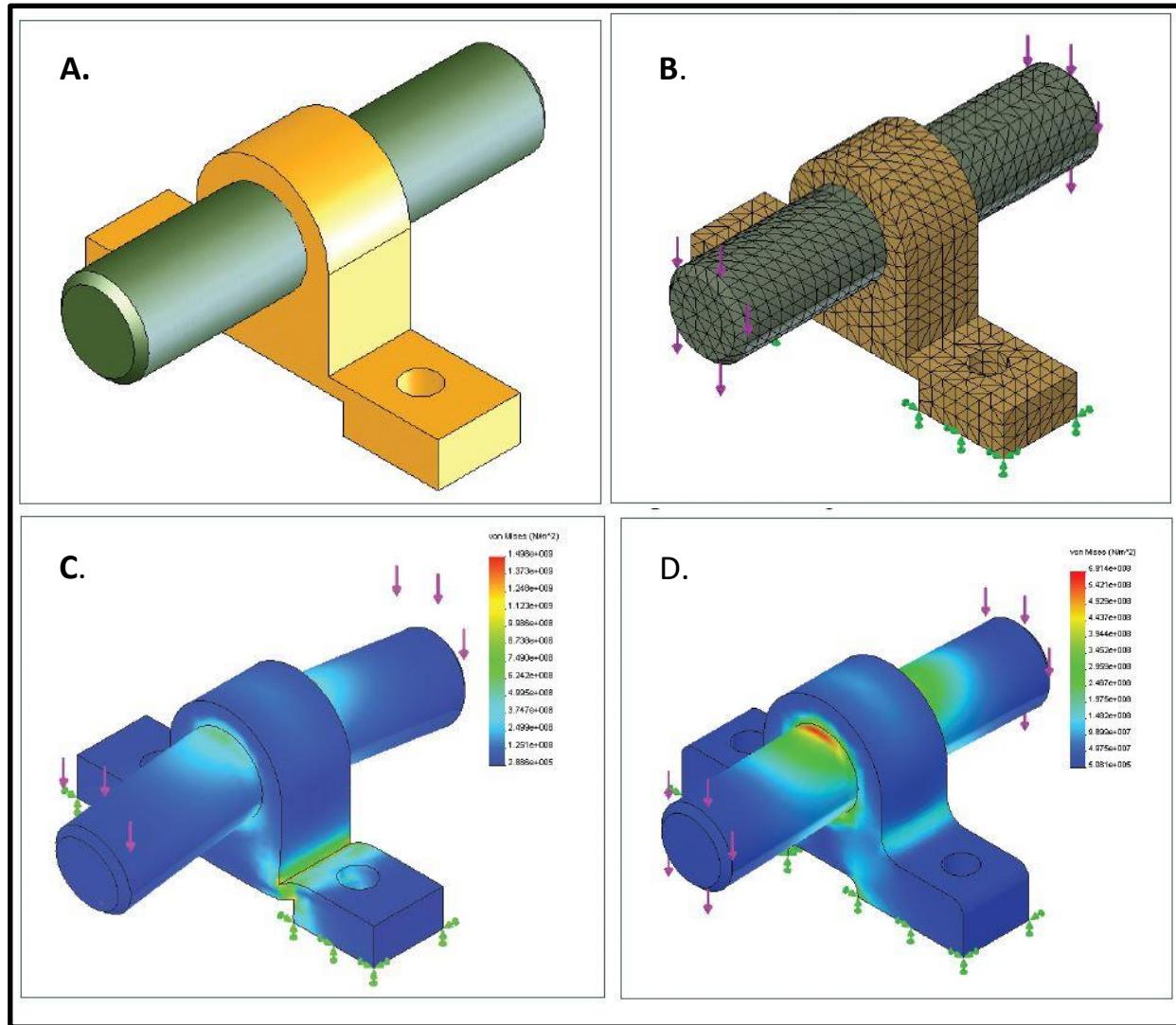
## **Class 9b:**

### **Team Project Analysis II**

9b.1 Finite Element Analysis (FEA) of One Part



**Figure 9b.1** The general steps in conducting a Finite Element Analysis (FEA) study of a part under a load consists of: 1. Build the part; 2. Create new study; 3. Assign material to the part; 4. Apply restraints; 5. Apply the force loads; 6. Create the mesh; 7. Run the FEA study; and 8. Document the results (stress, displacement, strain).



**Figure 9b.2** Recall in a recent SolidWorks lab that you conducted an FEA study of the pillow block. In this case, you used the forces on a rod to see where the stresses concentrated on the pillow block. (A) Assembling the two parts correctly. (B) Applying the restraints on the bottom feet of the pillow block, applying the downward forces at the ends of the rod, and creating a mesh. (C) Run the study and show where the stresses concentrate. (D) In this case, the pillow block was modified to lower the stresses.

## **Design Check #6:**

# **Team Project: Finite Element Analysis of One Part**

Before your team starts this Design Check #6 FEA study, refresh your collective memory by reviewing the SolidWorks lab of the pillow block FEA study (Lab 7.1 in the class workbook).

First, select one part from your project assembly device that you are curious about how a load force (stress) would distribute through its mass. If you need to use a second part, first assemble it to the part of interest (see figure 9b.2). Research the typical loads the part of interest can be subjected to. Start a new FEA study of the part and select the material for the part. Next apply the restraints where the part will be fixed. Then apply the selected load force value and direction on the part where the load should typically be placed during normal operation. Next mesh the model to start the analysis. Then run the study to get the results (stress, displacement, and strain). You can try different values for the load and re-run multiple studies as your team deems needed. But be sure to save images and data of the results for each case. *The main goal is to see how the load force distributes through the mass of your part.*

For the design check #6 submission, include the following:

1. The standard team project cover sheet.
2. A 1-2 page typed discussion of how you set-up the study, the value(s) used for the inputs, and the steps taken to get the results. Include the results your team chooses and write your interpretation of the stress distribution through the part. Be sure to use the correct units.
3. Image(s) showing the stress distribution of your SolidWorks part for the various cases your teams wishes to show. As a rule of thumb, show no more than three cases (lowest load, median load, and highest load) based on your initial research. Also, include the origin of the model that was used for the FEA study, on the images.
4. Include the D.C. #6 grading form at the end of your report.

Team Name \_\_\_\_\_ Unique No. \_\_\_\_\_

## **Design Check No. 6 Grading Form**

Grade

### Cover Sheet (5 points)

1. Semester and Year
2. Project Title
3. Team Name and Logo
4. Team Members Names, Emails, Leader
5. Instructor Name and Section Unique No.

### FEA Study Set-Up and Run Discussion (10 points)

1. Rationale for Part Selected
2. Rationale for Material Selected
3. Restraints Application Points
4. Load Application Points
5. Input Values Used for Loads
6. Results Values Listed
7. Interpretation of Results
8. Discussion Linked to Images
9. Font and Formatting Style
10. Writing and Grammar

### Images Showing FEA Results (10 points)

1. Image of Results #1
2. Image of Results #2
3. Image of Results #3
4. Orientation of Images
5. Size of Images
6. Quality of Images
7. Origin Depicted on Images
8. Use of Captions and Numbering
9. Font and Formatting Style
10. Overall Graphical Quality

Total Grade (25 points max.)