Simple SolidWorks Simulation Study

Simulating a simple structure mechanical analysis in SolidWorks 2017 step by step.

1. Open the SolidWorks file Analysis_1.sldprt
2. Go to tools menu bar and select "add – ins"
3. A simulation pan will show. Select both boxes from left and right next to "Solidworks Simulation"
4. Solidworks will install this new feature. It could take a few minutes before it can display the Simulation tab in the top part of the sketch screen.
5. A Simulation tab should now appear.
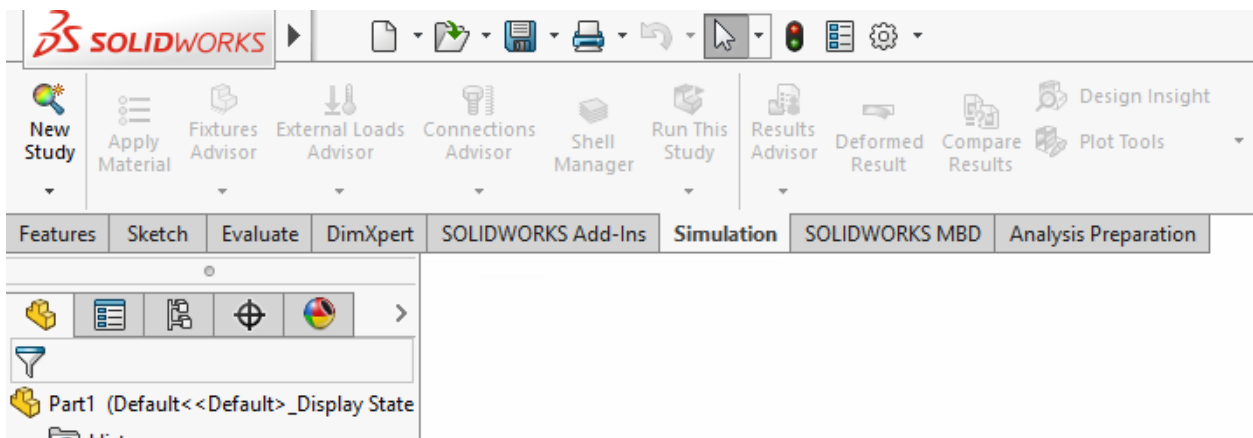
6. Whit the Simulation tab selected click on "New Study".
7. Name your study. Stress_Study_1 for example or select a prefix tittle like "Static" icon.
8. Click the green check mark to accept.
9. Now you can see the new study tab in the bottom of the screen.
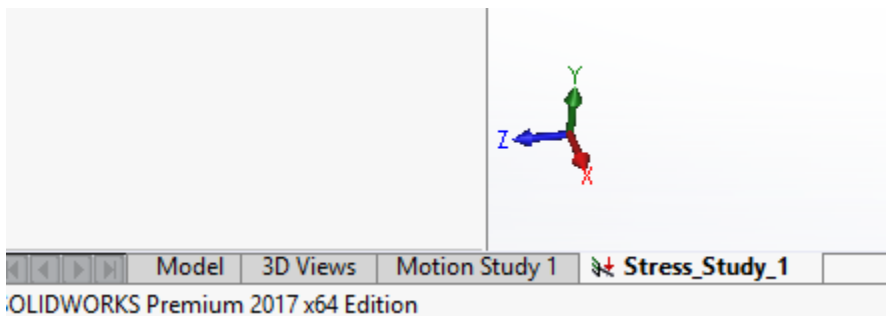
10. You can now see also a simulation tree in the right of the screen.
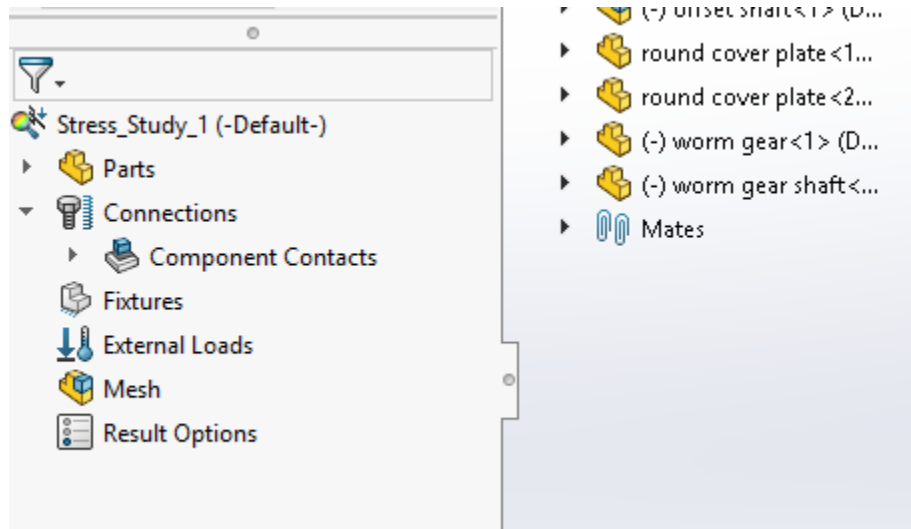
11. Right click on "Parts" And select "Apply / Edit Material".
12. Select Plastic from the list and then select "ABS".
13. This material is close to the plastic you can get when printing a 3D object with a 3D printer.
14. Use any material you want for your project.
15. Once the material has been selected a green check mark will appear on the part icon in the simulation tree.
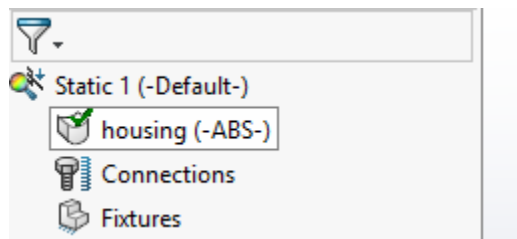
16. Now we must fix the part to a surface so the support can be made and the part won't move. This surface will be fixed.
17. Right Click on "Fixtures" and select "Fixed Geometry". Rotate the model so you can select the surface you want to be fixed. Just click on that surface then click the green check mark to accept.
18. Now we will apply a force to the model.
19. Right click on "External Loads" and select "Force".
20. Select the surface of the object that will take the force applied. Rotate the object if needed. Then type the amount of the force in Newtons.
21. Click the green check mark to accept.
22. This step will define the amount and direction of the force applied in this simulation. Later on this will define how your structure can handle that amount of stress.

23. To Run the simulation, we need to define the mesh for the structure we need to analyze. The mesh will divide the object in many vectors. Depending on how many vectors you want the object to be divided the longer the time the software will take to analyze the study.
24. Right click on "Mesh" and select "Create Mesh". By default, the mesh is midway between Coarse and Fine. You can change it to vary the definition of your study. Start with the middle point for now.
25. Remember a too Fine analysis will take longer to finalize depending on how complex your design is.
26. Accept it by clicking on the green check mark.
27. Now right click on the first icon on the simulation tree "Stress_Study_1" and select "Run"
28. This will take several minutes depending on how complex the simulation is.
29. Once this finishes you will be able to add or remove parameters like definitions (Units, Newtons or Pascals). Add motion to the simulation to see how it can deform or depending how much force you apply to it break if that is the case.
30. Remember that the Blue color is less stress and Red color is the most stress or breaking point due to a lot of stress.
31. Please make sure you can add and run different features to the simulation. This will be mandatory for your final project.

Thanks, and enjoy the simulation.

Instructions made by:

New York City College of Technology

Department of Computer Engineering   EMT2480L   Spring 2017

Prof: Mauricio Cardenas