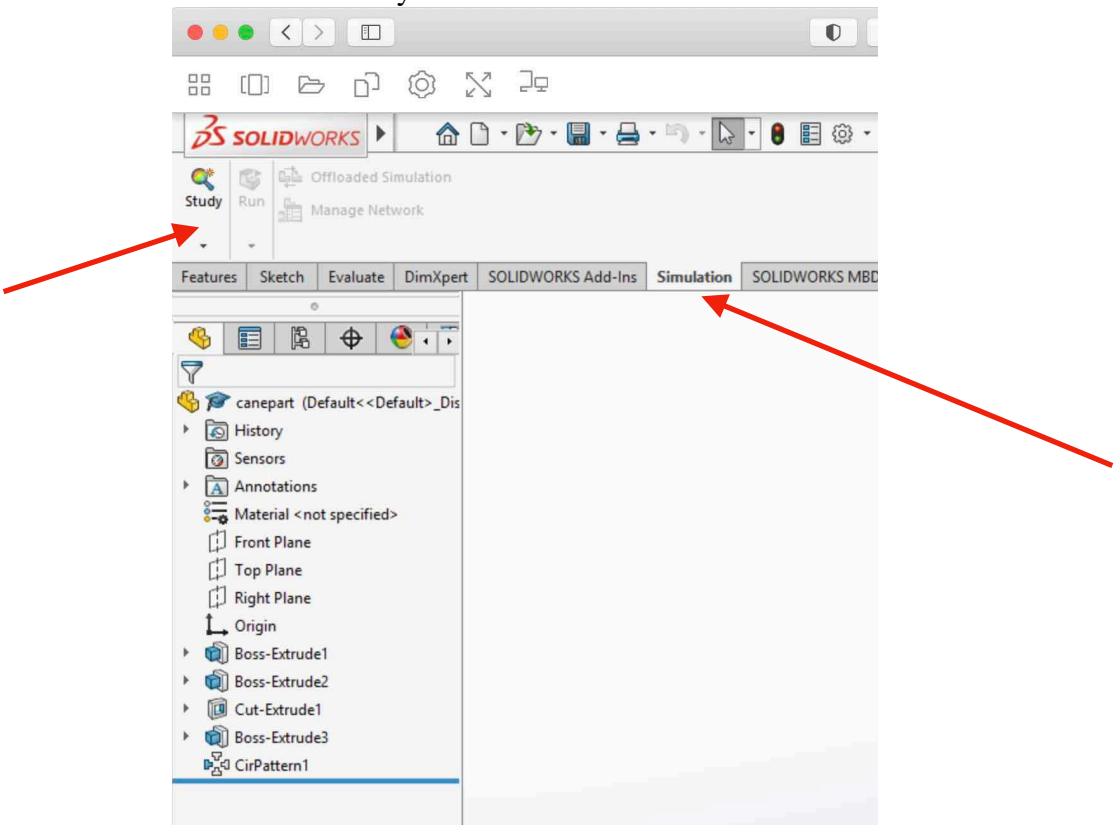
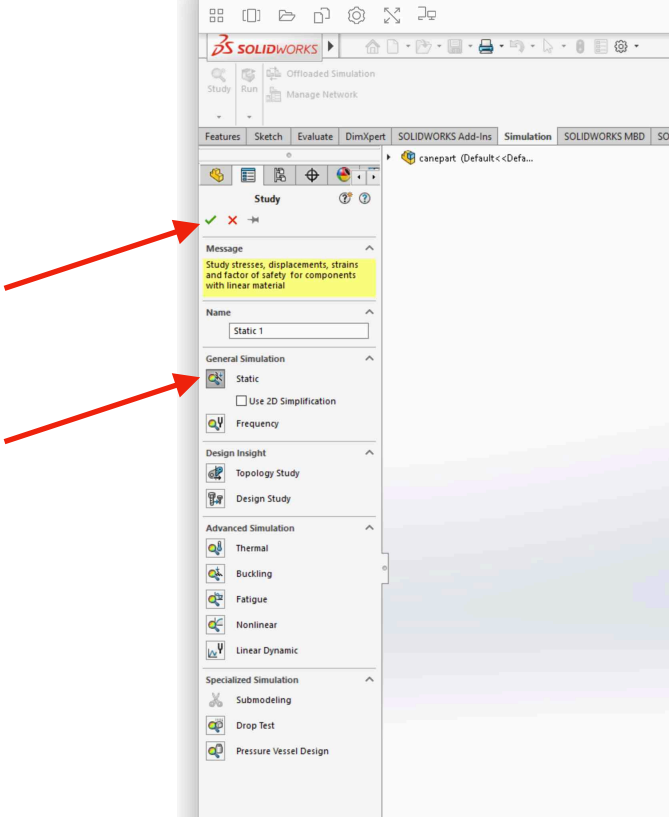
Solidworks Simulation
BME 200

Procedure

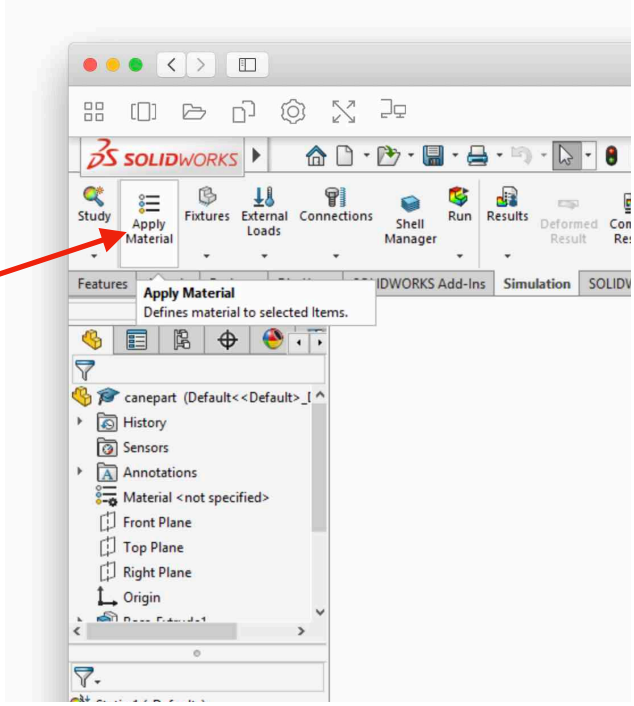
1. Left-click on Simulation then Study.



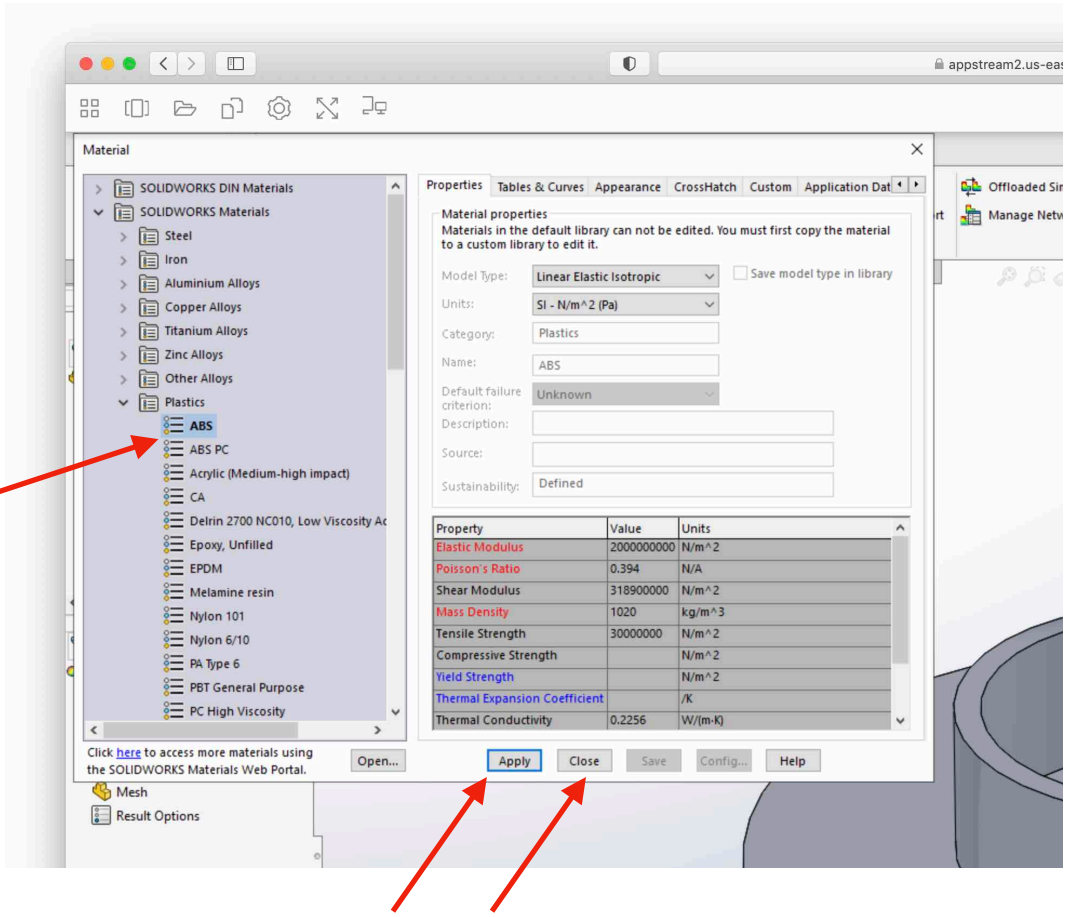
2. Static is selected by default. Confirm that it is selected and then click the green check mark.



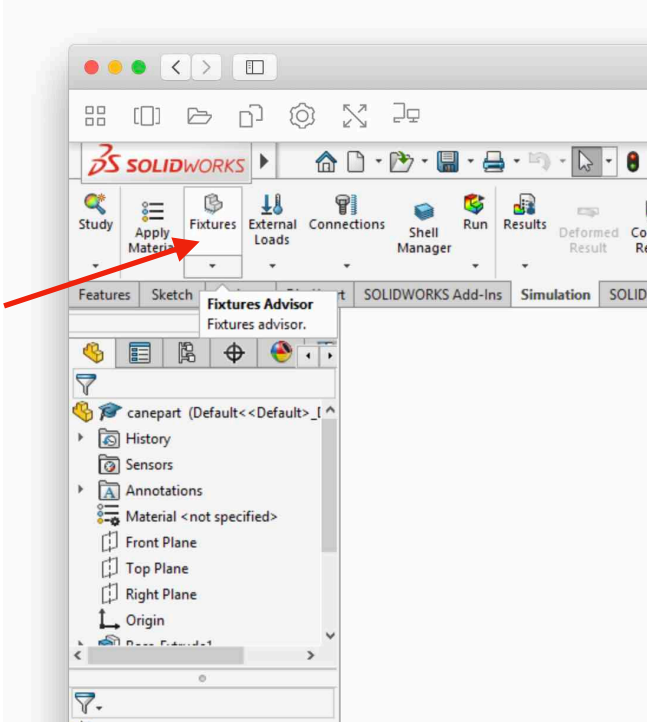
3. Click Apply Material and be patient! It may take ~10 seconds for the next window to pop up.



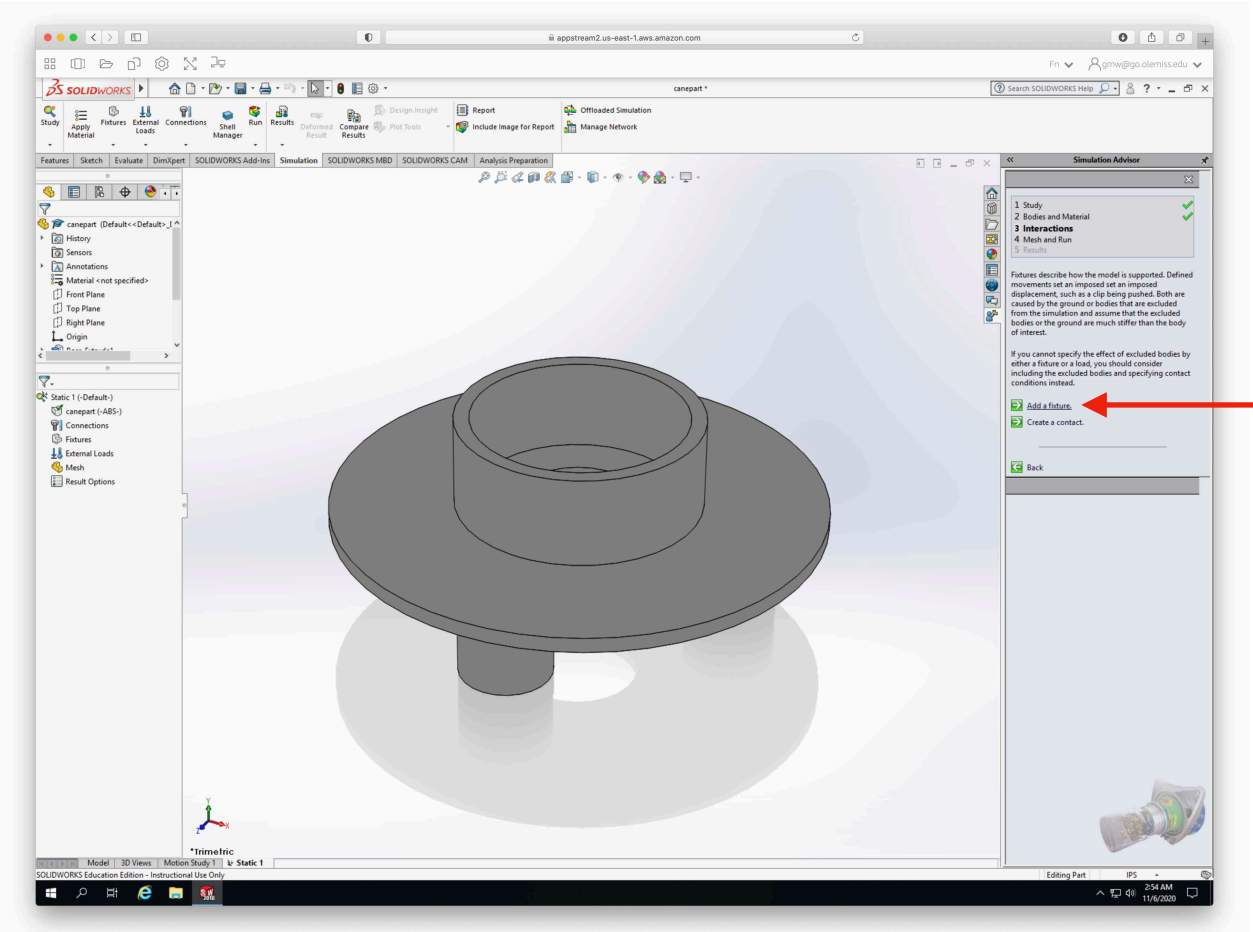
4. Choose the material ABS, under Plastics. Click Apply. Then click Close.



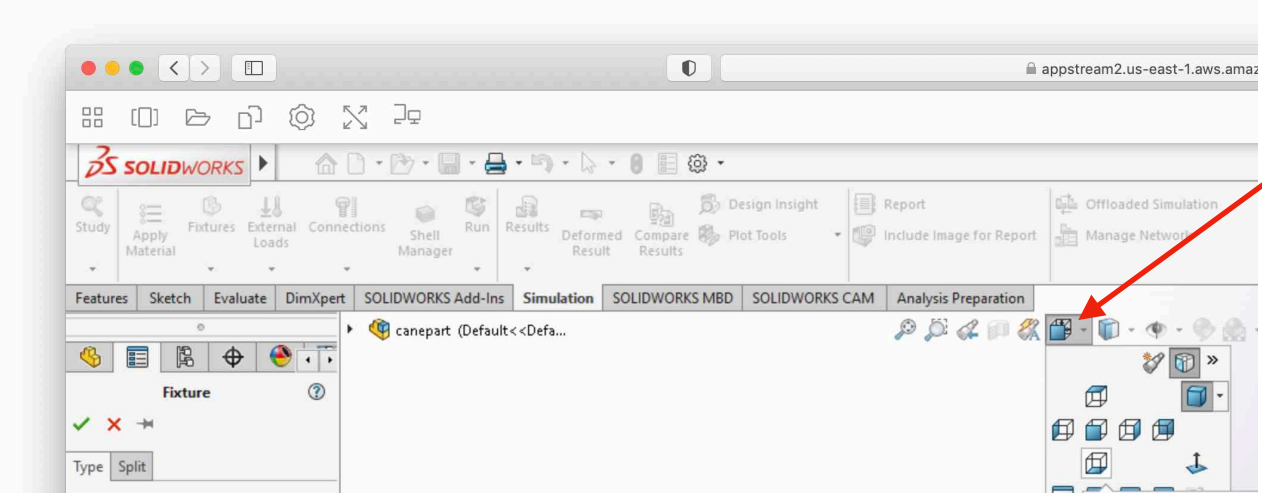
5. Next, click Fixtures. A fixture is one or more surfaces on your model that will remain fixed when the load is applied.



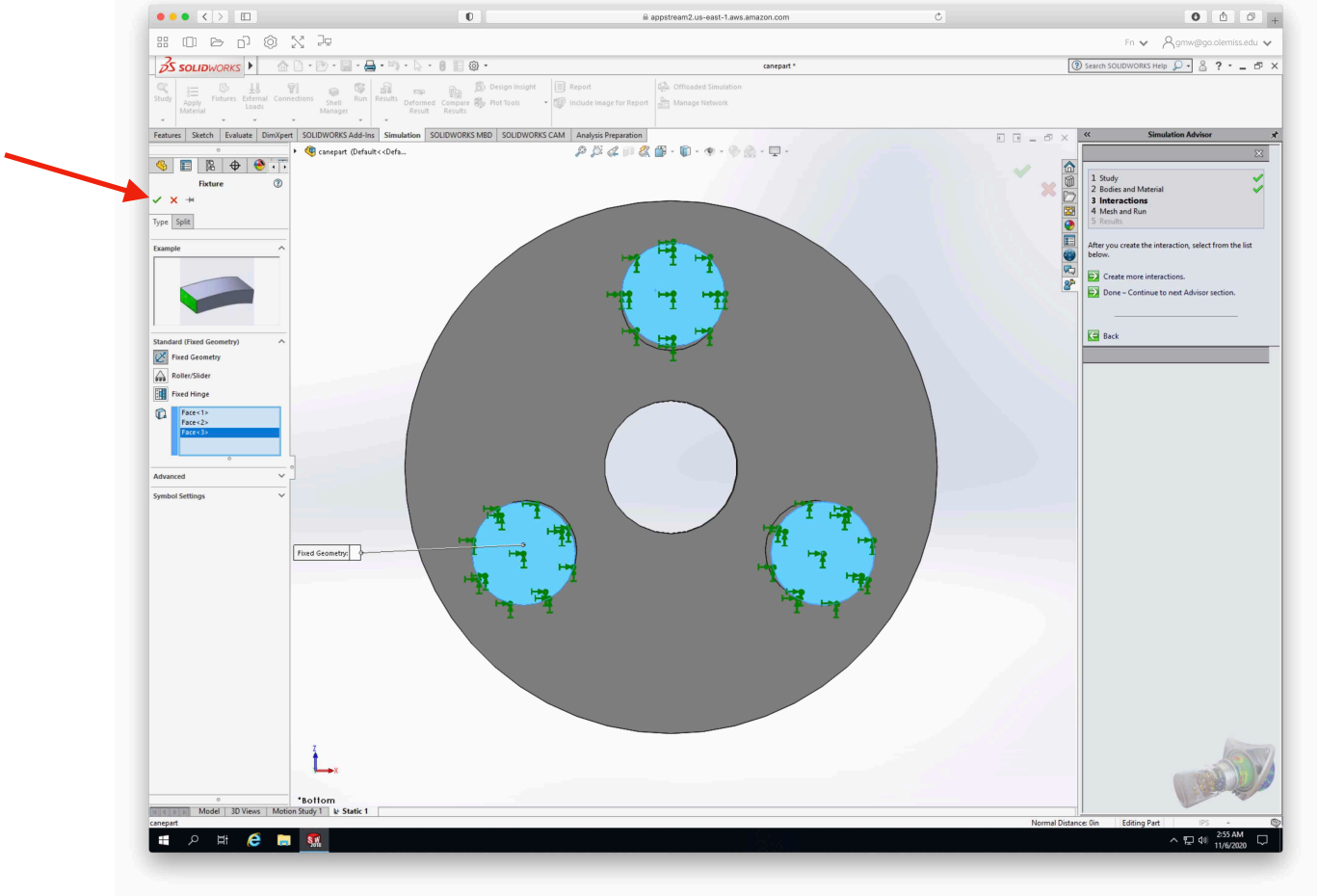
6. Then click Add a fixture on the right side of the screen.



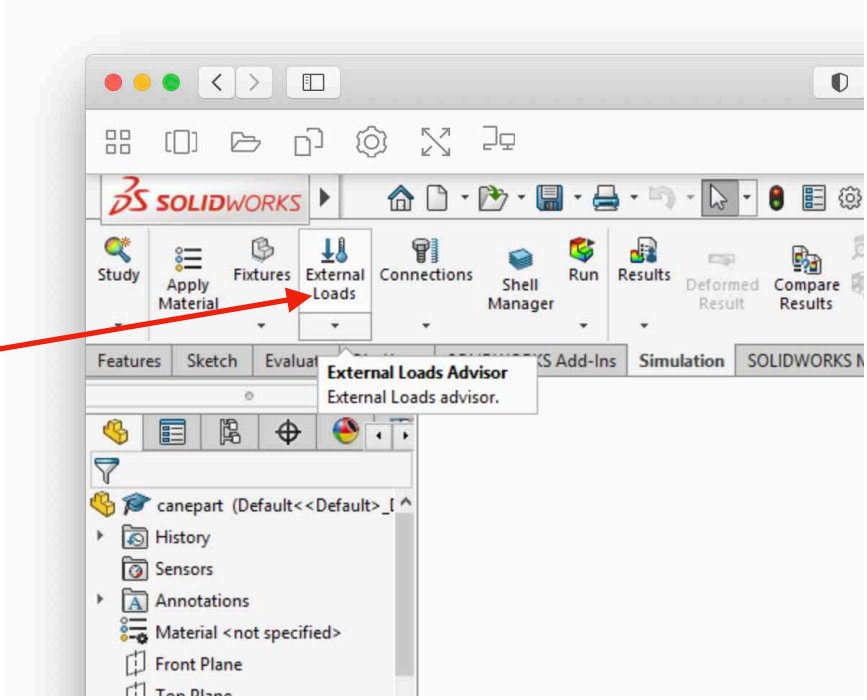
7. Flip the part so that you can see the surfaces you want to add fixtures to. There is no "correct" view. Use whichever one gives you the best angle.



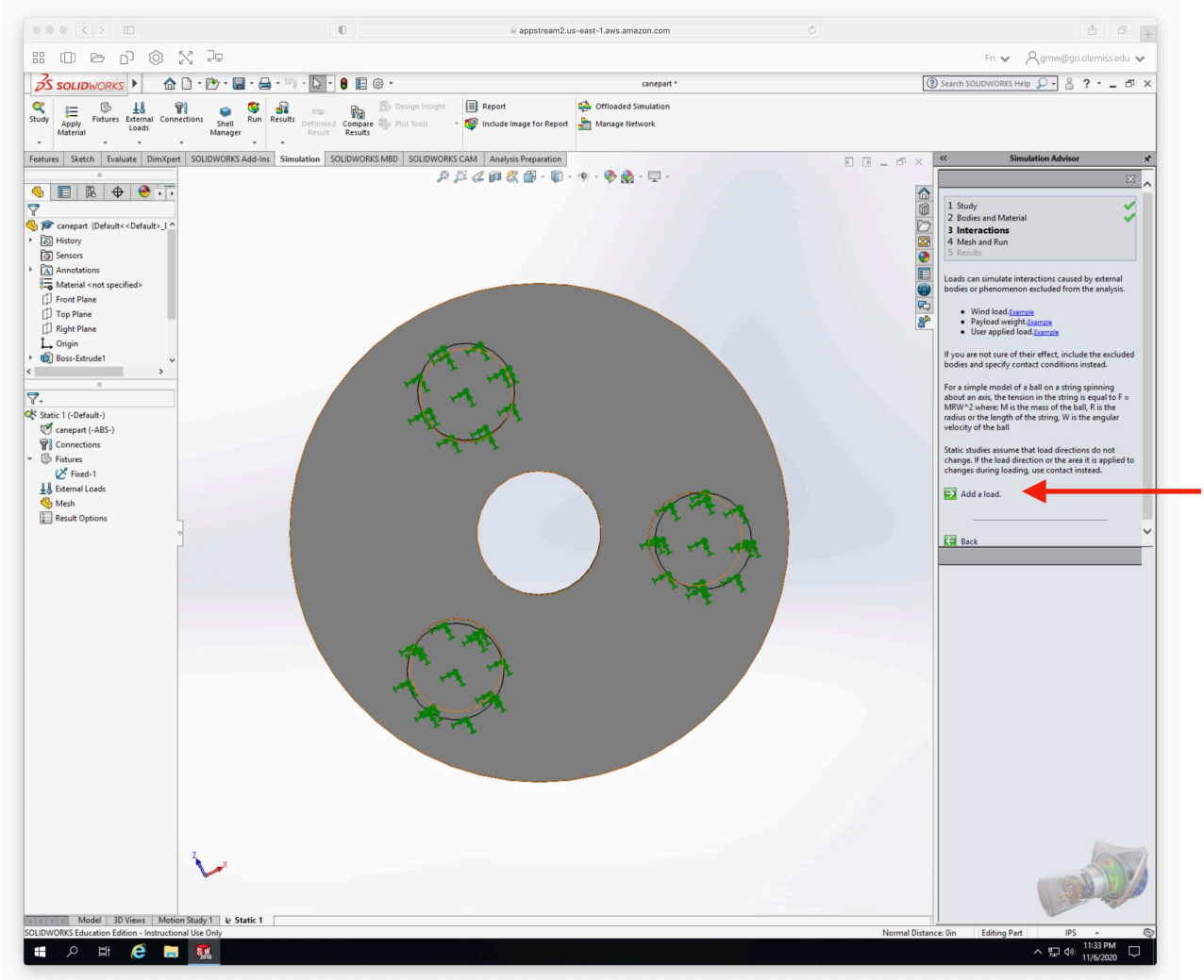
8. Select the surfaces you want to remain fixed. A list of the surfaces (faces) you select will appear on the left. The structures in the diagram will turn blue. Once you are done, click the green check mark.



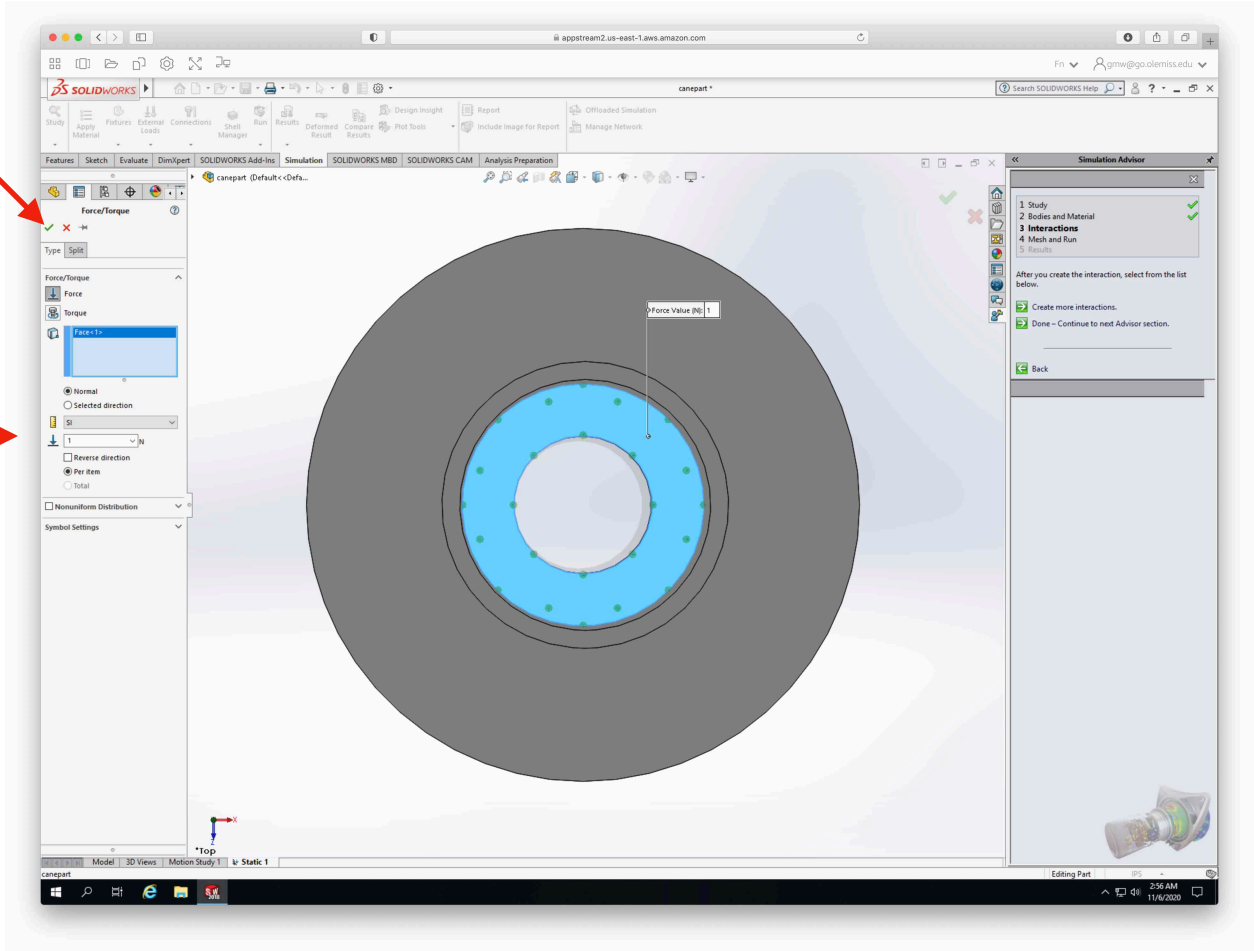
9. Next, click External Loads.



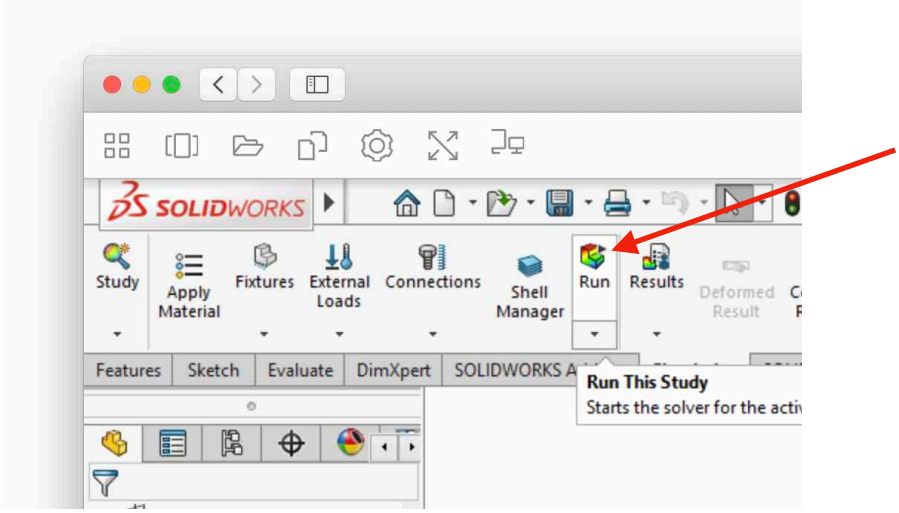
10. Then click Add a load on the right side of the screen.



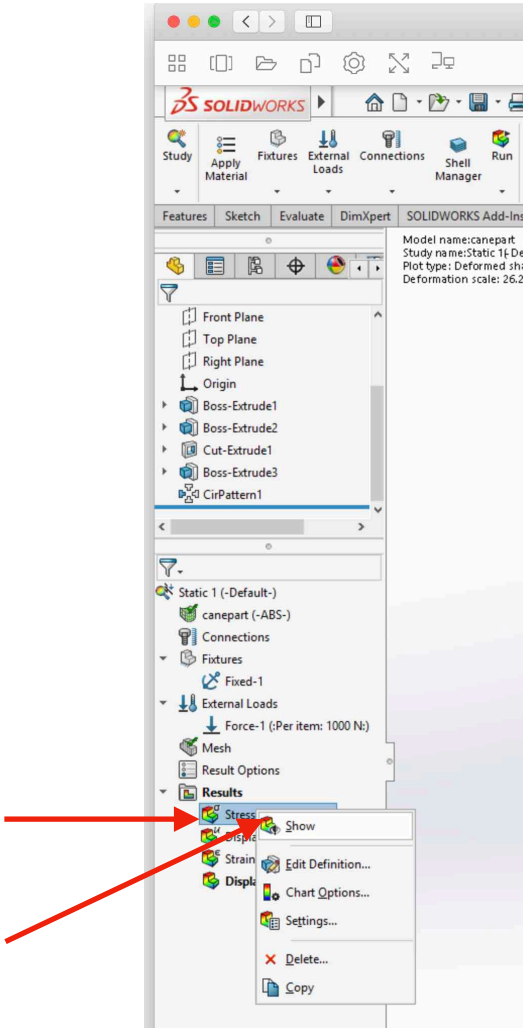
11. Arrange your part so that you can see the face on which you want to apply the force. Then, click on the face. Enter the amount (in Newtons) of the force in the blank. Then click the green arrow.



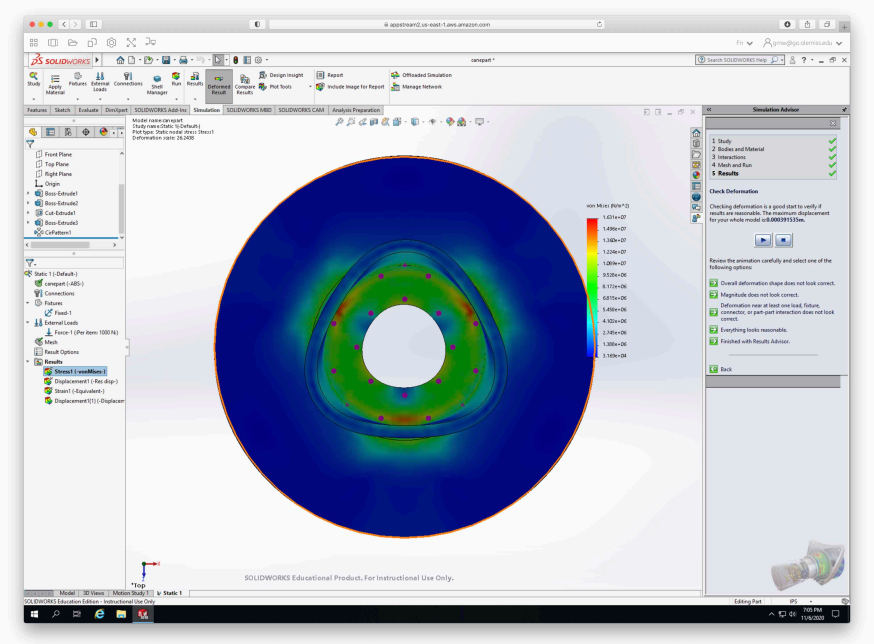
12. Click Run.



13. Once the simulation is done, right-click on Stress1 then left-click on Show.



14. Use File->Save As PDF to save a PDF of your stress results.



Summary

These are the steps you performed to run the simulation:

1. Specify the part
2. Specify the type of simulation (static load)
3. Specify the materials for the part (ABS)
4. Specify the fixtures. These are the parts that remain fixed while the load is applied.
5. Specify the external loads. These force(s) applied to the part.
6. Run the simulation.

NOTE! If you want to adjust a dimension and re-run a simulation, you do not need to go through all these steps each time. Simply adjust the dimension of the part you are interested in and then run the simulation again.