

Solidworks tutorial



**simulationXpress analysis
wizard**

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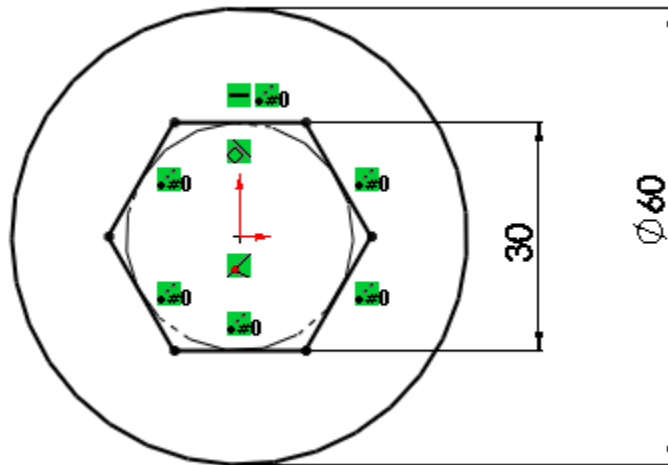
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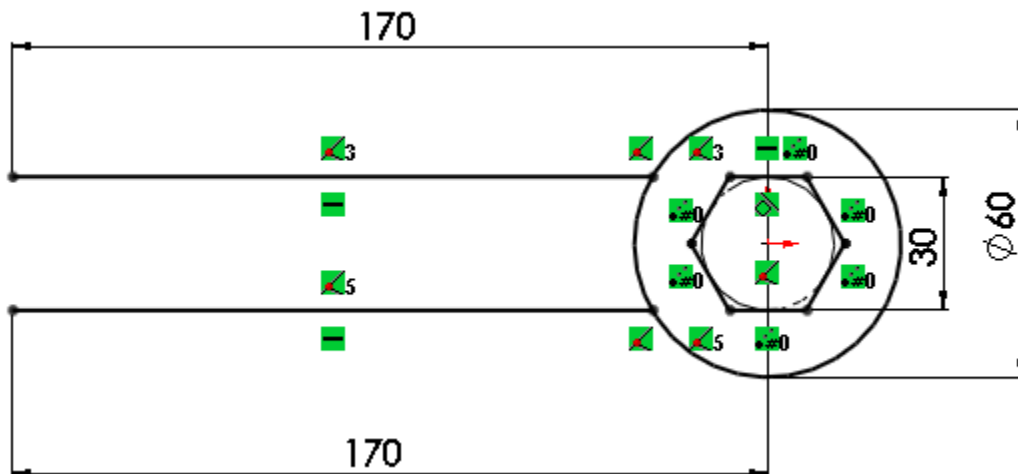
This time we are going to use solidworks simulationXpress analysis wizard. Here we just want to show the performance of this solidworks tool so we keep it simple. We will design a wrench and we will analyze it by putting some load. So let's begin with the designing of the wrench.

Create a new part document in solidworks and save it by the name of wrench. Select the top plane and create the below sketch on it:

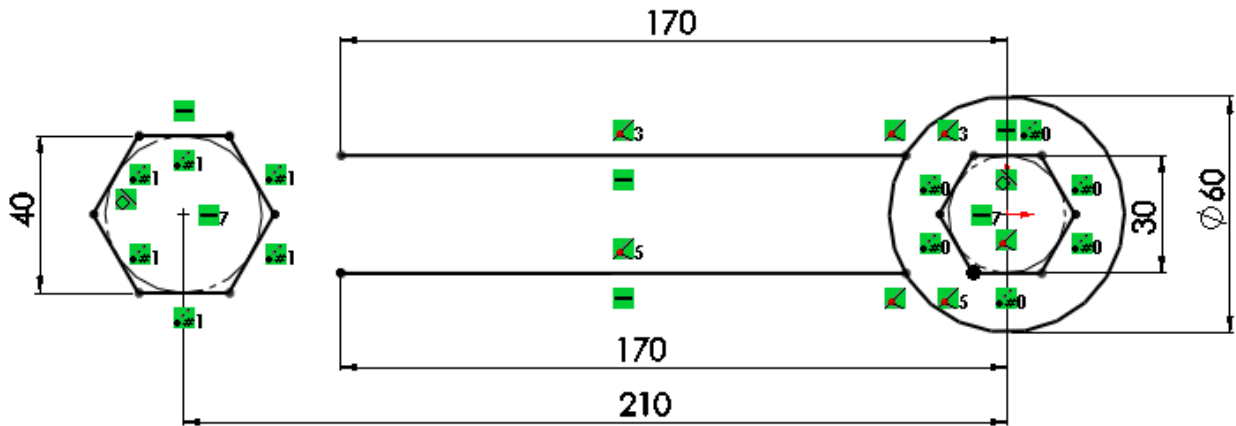
First draw this polygon on the origin point with a circle around it



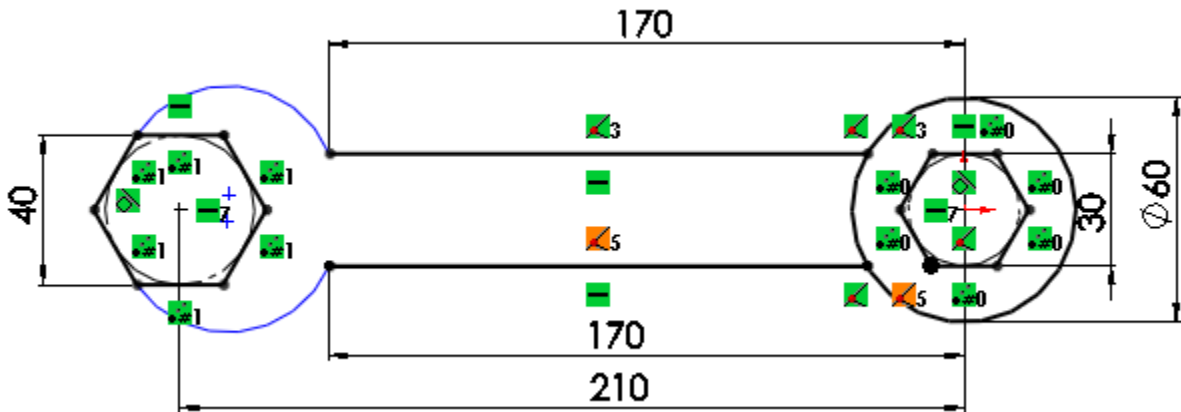
Now add below lines



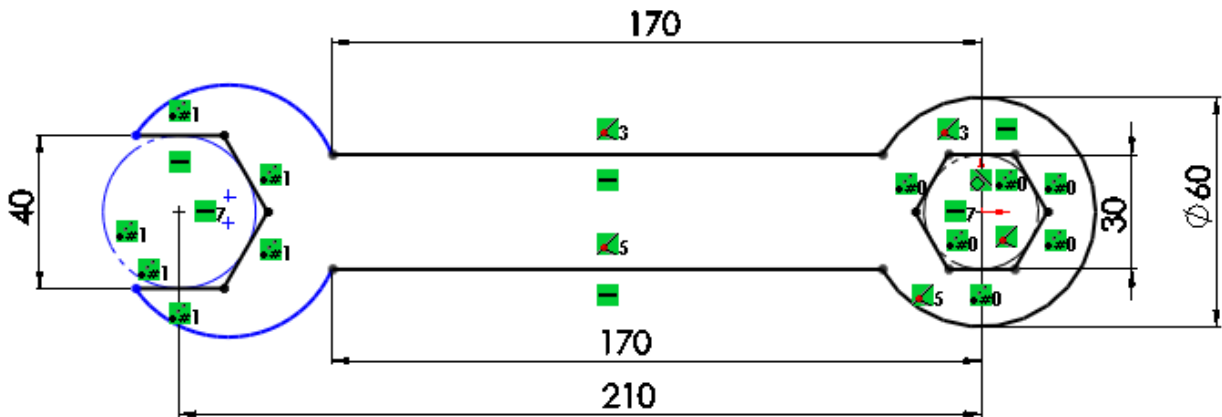
Now add the opposite head of the wrench. First draw another polygon



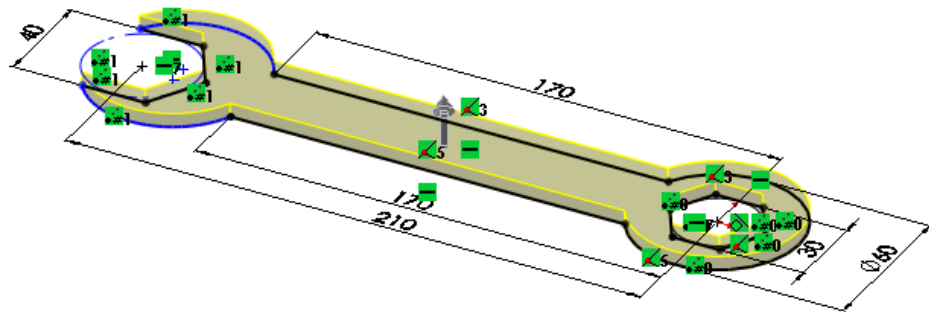
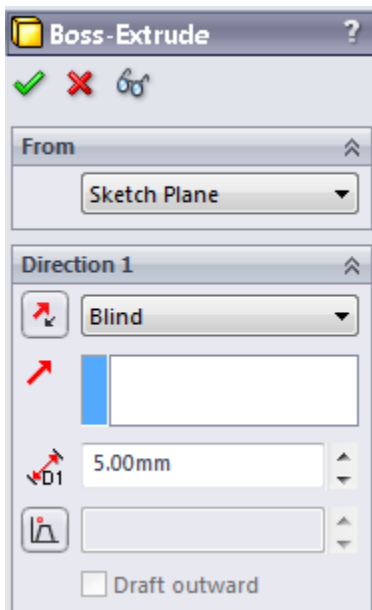
Now add two arcs like below. No need to be exact in this step we just want to finish the wrench and start simulation



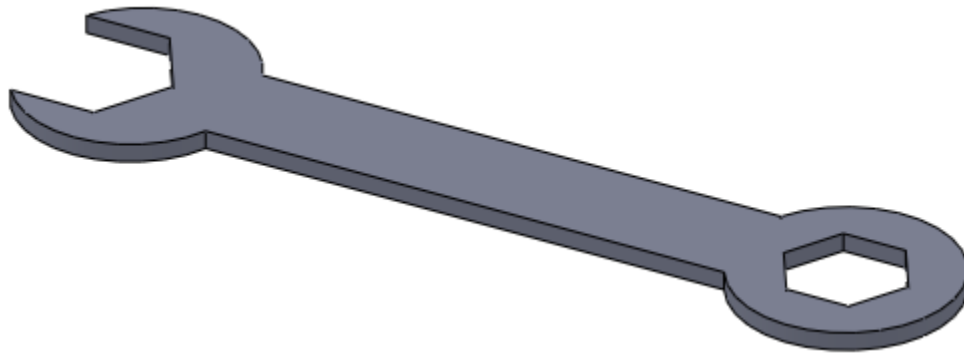
Trim the unwanted lines



Now extrude the drawing



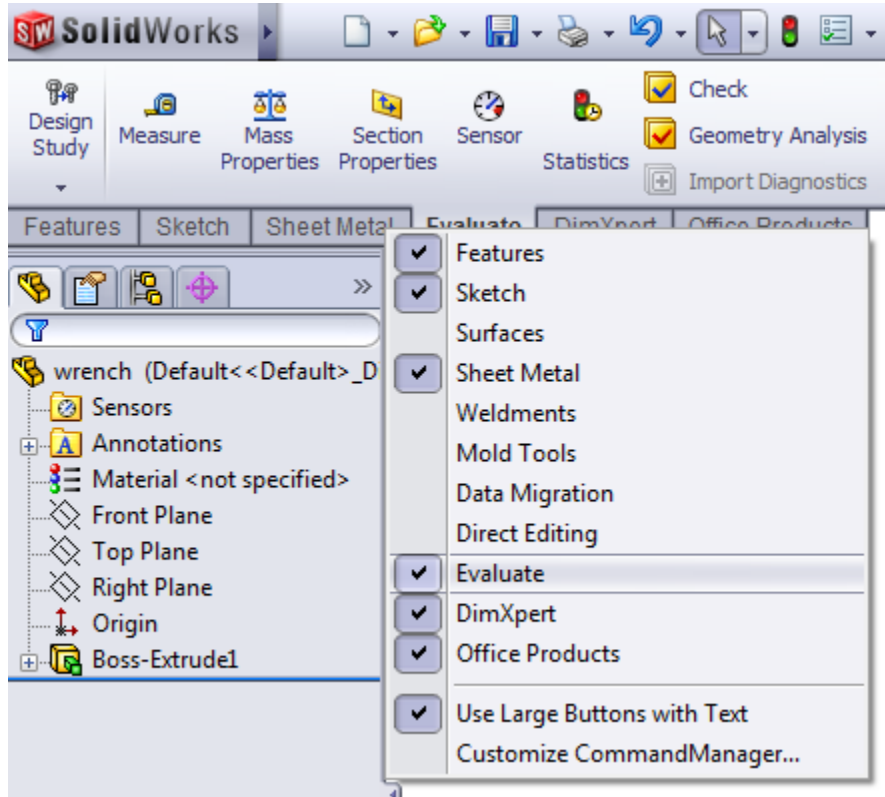
Now we have our wrench and we can start our simulation analysis.



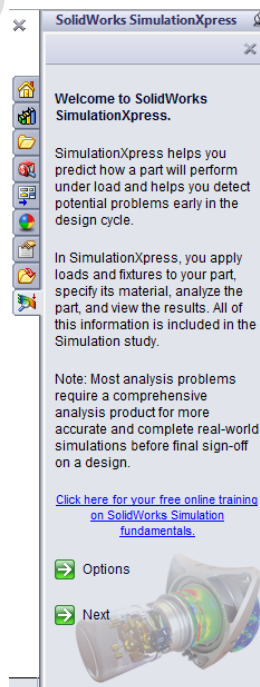
Solidworks simulationXpress performs in six below steps which we are going to do step by step:

1. Fixtures
2. Loads
3. Material
4. Run
5. Results
6. Optimize

Ok let's simulate. While the wrench part is open navigate to the "evaluate" tab, if the tab is not visible right click on any tab and check the evaluate tab to make it visible.



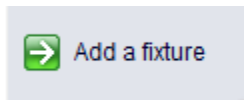
in evaluate tab, click the "simulationXpress analysis wizard" button and the simulation column will open on the right



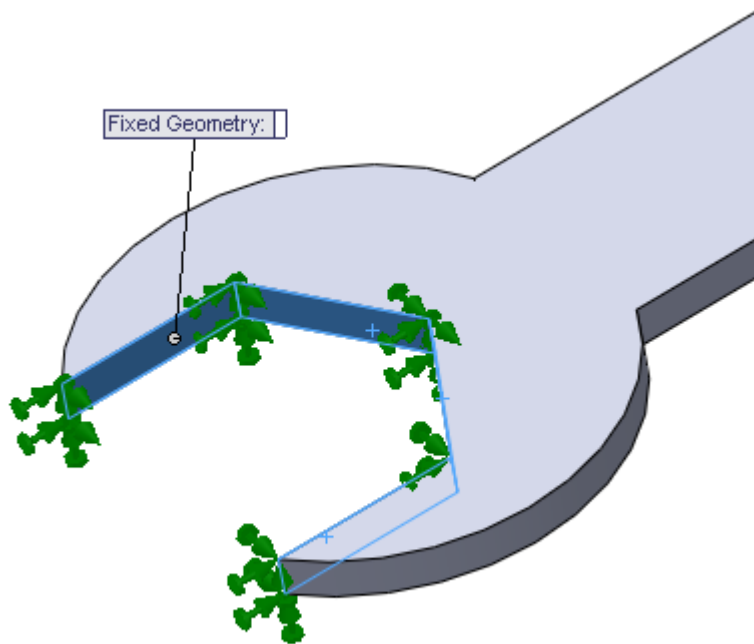
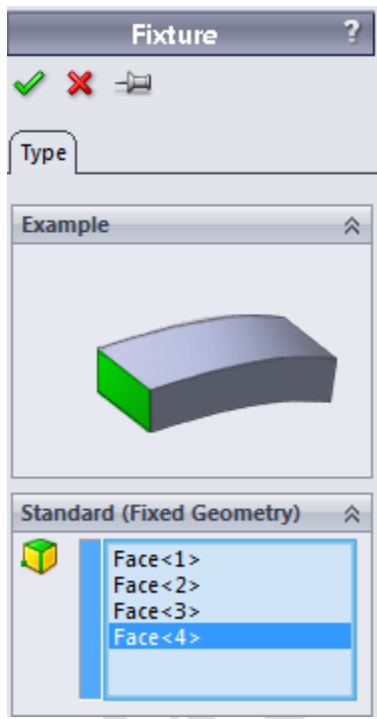
Feel free to read the note provided by solidworks which will help you to understand the process.

Click the next button below the notes.

In this step we need to define the fix sections in our simulation so click add fixture button.



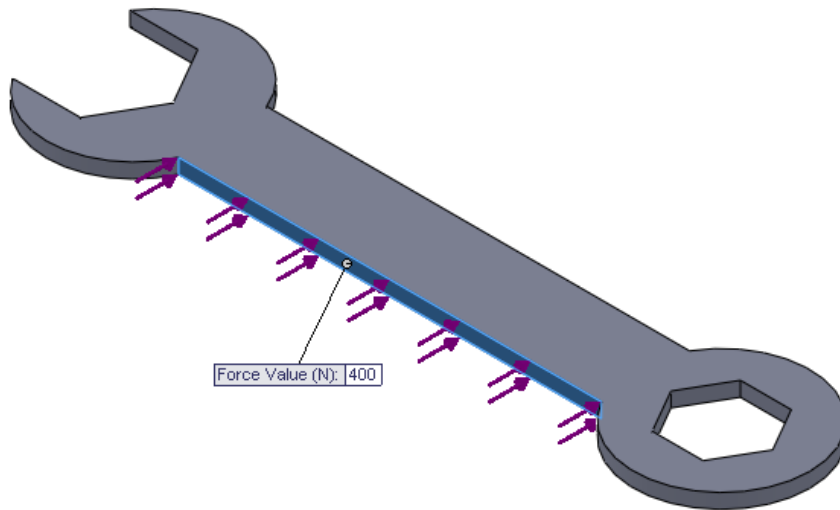
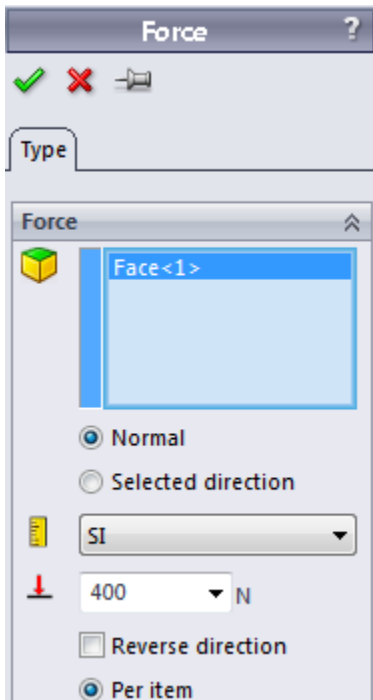
The fixture window appears and now we need to select below faces as fixtures and click ok



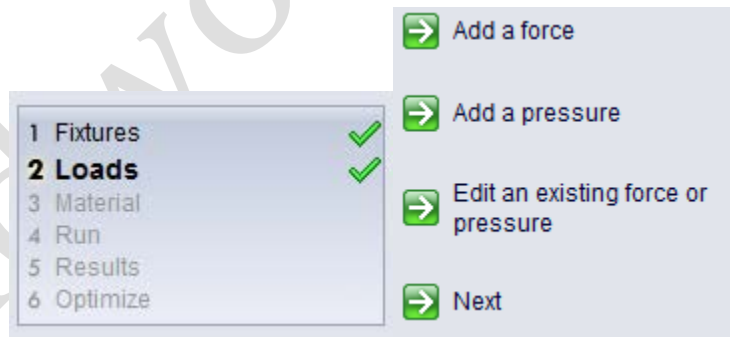
Click next to define loads in next step. Now the fixtures are checked.



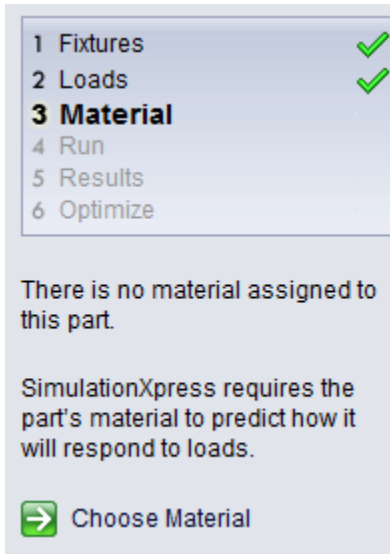
Click “add a force” button and force window appears on the left. Click the below shown face to define the force direction. Set 400 N as the amount of force Then click ok.



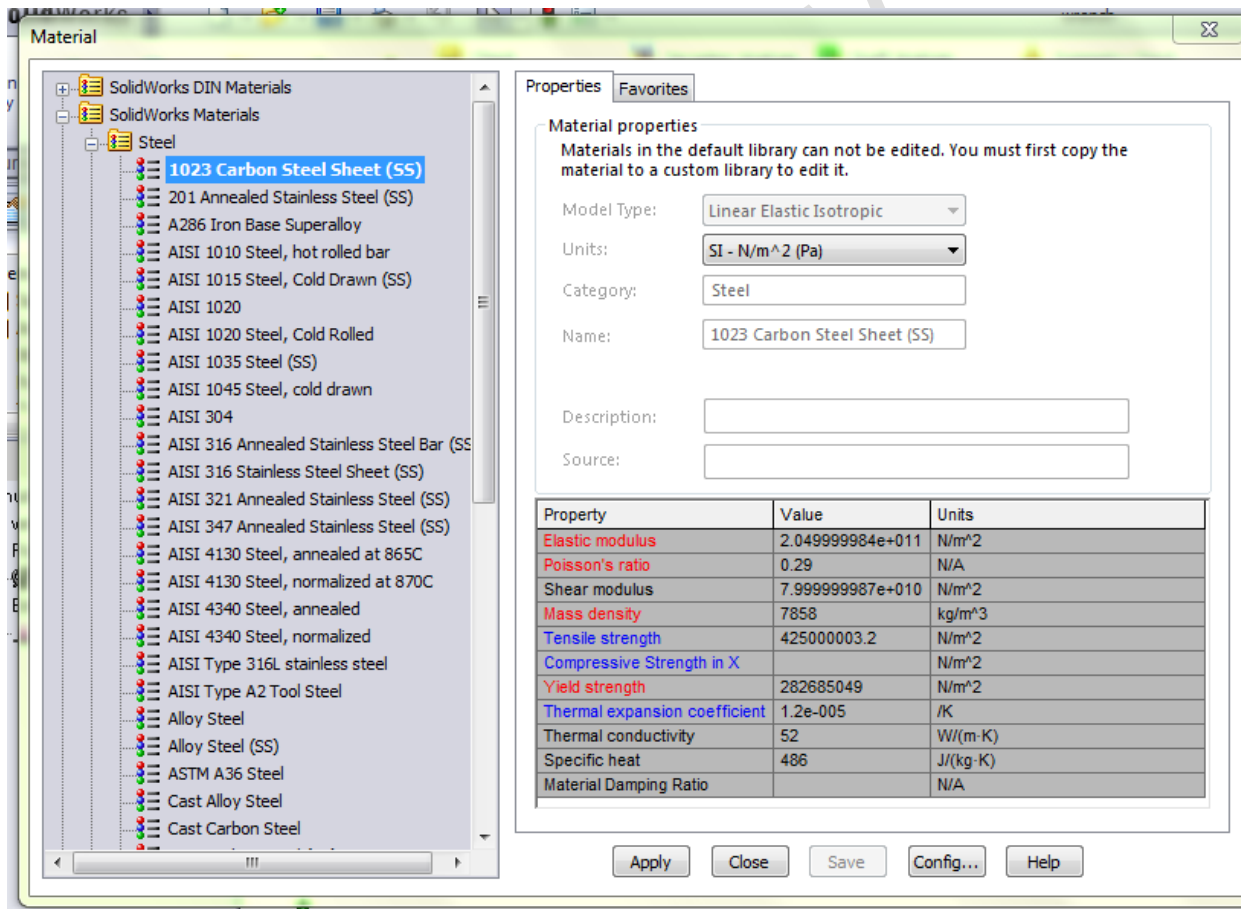
Now the loads are checked and we can click next to define the material



Click choose material button



Material window appears. Select 1023 carbon steel from steel category then click apply and finally click close



Now material is checked now. Click next in simulation window. Then click run simulation

1 Fixtures ✓
2 Loads ✓
3 Material ✓
4 Run
5 Results
6 Optimize

The material assigned to this part is:

1023 Carbon Steel Sheet (SS)

Young's Modulus:
2.05e+011N/m²

Yield Strength:
2.82685e+008N/m²

➔ Change material

➔ Next

1 Fixtures ✓
2 Loads ✓
3 Material ✓
4 Run ✓
5 Results
6 Optimize

Your model is ready to solve!

You can solve with the default settings or adjust them to better suit your needs.

➔ Change settings

➔ Run Simulation

It will take some seconds for solidworks to calculate then our wrench will start to move like an animation. If you are satisfied with part deformation you can click the “yes continue” button to see the results

1 Fixtures ✓
2 Loads ✓
3 Material ✓
4 Run ✓
5 Results ✓
6 Optimize

Examine the animation of the part's response to verify that the correct loads and fixtures were applied.

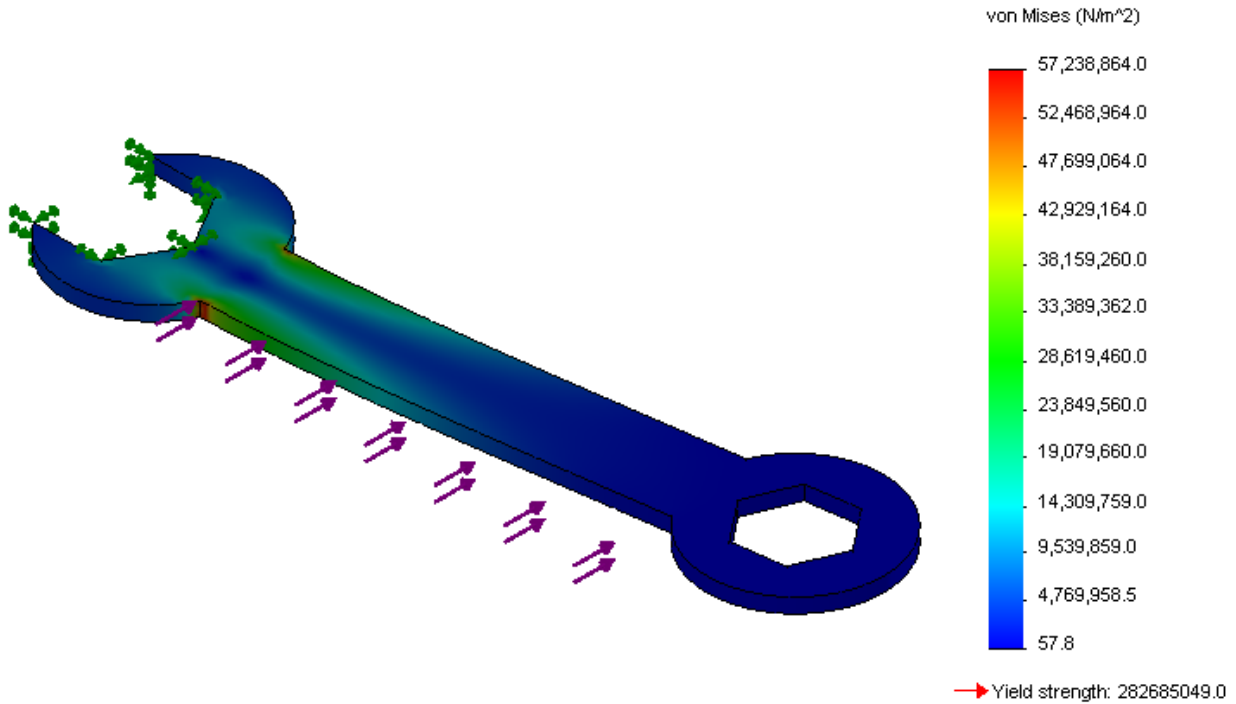
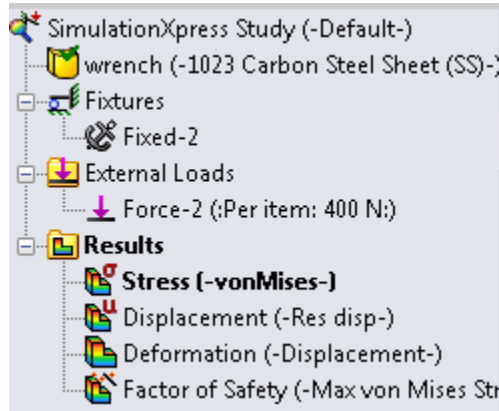
Warning: If the loads and fixtures are incorrect, the results of the analysis will not be accurate.

▶ Play animation
▣ Stop animation

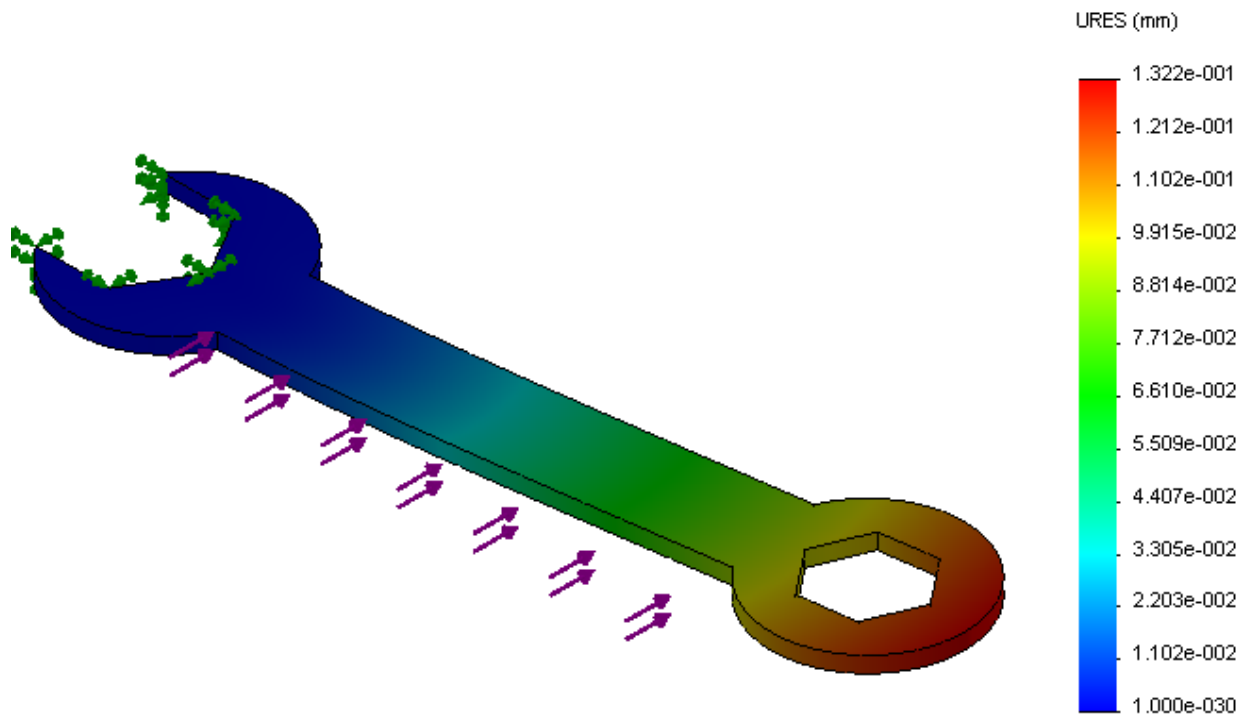
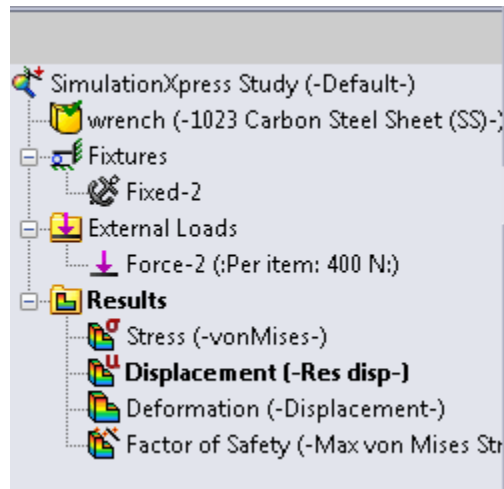
Does the part deform as you expected?

➔ Yes, continue
➔ No, return to Loads/Fixtures

Now you can check the results on the feature manager design tree. For example double click the stress to see the analysis results on the screen

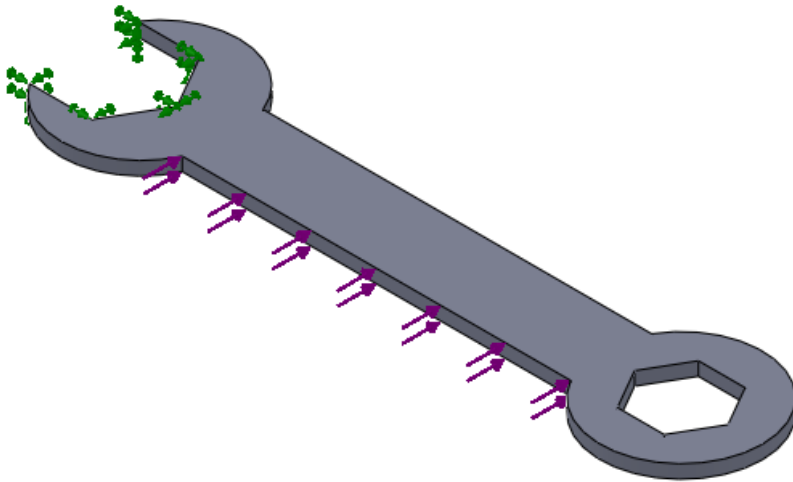


Or click displacement to see the movement analysis of the part



Take your time and examine the results for a while to see how this tool works. When you done with the results you can click "done viewing results" to go to the next tab.

At the final step you can generate a HTML report or edrawing file from your simulation analysis



1 Fixtures	✓
2 Loads	✓
3 Material	✓
4 Run	✓
5 Results	✓
6 Optimize	

Saving a report of your results ensures the information is well documented for future work on this or similar projects.

Choose between these two report methods:

Click next. Here you can optimize your process which for now we don't want to.

1 Fixtures	✓
2 Loads	✓
3 Material	✓
4 Run	✓
5 Results	✓
6 Optimize	

Optimize Your Design

SimulationXpress can identify the optimal dimension for most features in your SolidWorks model based on your simulation results.

Would you like to optimize your model?

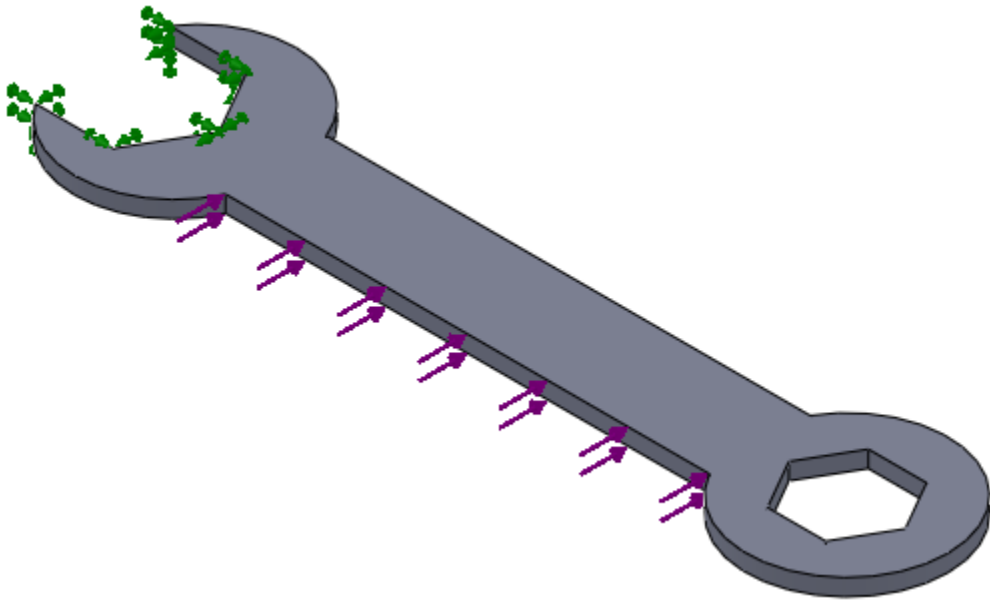
Yes

No

That's it. Congratulations you just finish the simulation now you can analysis more complicated parts by this tool in your future works.

1 Fixtures	✓
2 Loads	✓
3 Material	✓
4 Run	✓
5 Results	✓
6 Optimize	✓

You have completed the SimulationXpress Wizard.



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