**Demo 1 Step-by-Step: Homework 6 Problem 5**

The problem in this SolidWorks simulations demo is from the bending stress homework 6 problems as below:

Diagram

Description automatically generated

Compared to this problem, students should determine that, given an even distributed load (w) of 1000 N/item in a SolidWorks model of the above system, would the maximum stress exceed the of 10 MPa?

To answer this question, we have created a SolidWorks model with dimensions exacting that of the above problem with a few assumptions such that the model is a 3D rather than a 2D problem as in here. Using SolidWorks statics simulations, we can determine the stress throughout the beam.

1. Download the HW6P5 folder and unzip the contents (Or open the file using Citrix). You should see a model below.
2. Make sure that the Simulations tab is visible in your SolidWorks window. Right-click the toolbar at the top of your SolidWorks, go to the Tabs option and ensure that SOLIDWORKS Add-Ins is checked.

Graphical user interface, application

Description automatically generated

1. Go to the Simulation tab in your toolbar and select New Study at the top left of the screen to open a panel that allows you to define simulation type and parameters.

Graphical user interface, text, application

Description automatically generated

1. Make sure Static is selected and go with the default settings by clicking the checkmark at the top. After confirming the study, you will see the screen below that allows you to define component interactions, connections, and external loads, and generate a mesh.

Graphical user interface, text, application, email

Description automatically generatedGraphical user interface, text, application, Word

Description automatically generated

1. To tell the software that our fixtures at the bottom of the object will remain in place, click on the down arrow at the Fixtures Advisor tab at the top and select Fixed Geometry. Then, click on the bottom of your hinge support and rolling support as below. Make sure that both faces are selected in the blue box.

Graphical user interface

Description automatically generated with medium confidence

Select the rolling support.

Graphical user interface

Description automatically generated

Click the checkmark to confirm your fixtures.

1. To apply the distributed load, go to the External Loads Advisor down arrow and select Force, which will open the window below.

A screenshot of a computer

Description automatically generated with medium confidence

Select the top face to indicate that your force will be incident on the top of the object. Then, enter the distributed force value of 1000 N per each item (arrow) as below.

Graphical user interface, application

Description automatically generated

Click the checkmark to confirm your changes.

1. We need to select a material such that the simulation can run appropriately. Click the H6\_P5 icon and hover over Apply Favorite Material to All Bodies. We can just use Plain Carbon Steel for our analyses, but keep in mind that changing the material does influence the output data as different materials are more or less resistant to loads.

Graphical user interface

Description automatically generated

1. We now need to create a mesh necessary for Finite Element Method (FEM) analyses, which allows us to measure the simulation at certain nodes and interpolate between that data.

Graphical user interface, application

Description automatically generated

Under the Run, This Study dropdown arrow, select Create Mesh.

Graphical user interface, text, application

Description automatically generated

This panel will open when you create the mesh. You can change the mesh density to obtain a finer (will take more time to run) or coarser mesh, which largely depends on your object morphology. We can just go for the default and click the checkmark.

1. We are now ready to run our study.

Graphical user interface, application, Word

Description automatically generated

Click Run This Study to run the simulation you’ve developed.

This is how the simulation results should look.

**Graphical user interface, application

Description automatically generated**

**We will now analyze our results to determine whether the stress along our beam ever exceeds 10 MPa.**

1. Right-clicklick Stress1 (-vonMises-) under the Results tab at the left panel and select probe.

Graphical user interface, application

Description automatically generated

Once the probe function is selected, a panel will open up to the left that will allow us to sample our object for values such as displacement and stress.

1. In the Options section, select On selected entities, to sample for stress along the edge of the beam. Then, click on the edge of the object, which should show up in the blue box as below.

Graphical user interface, application, Word

Description automatically generated

You will see values attributed to each node below the blue box. If you wanted to extract this data, you could copy and paste these into excel.

1. Graph the resulting probed edge by scrolling down to the Report options below the summary. Click on the plot option, which will generate a plot of parameterized distance vs. the computationally derived stress values.

Graphical user interface, application

Description automatically generated

1. After clicking Plot, you should see the graph below, which proves that the stress does not exceed 10 MPa (or 10 \*10^6 Pa).

Chart, line chart

Description automatically generated