**Demo 2 Step-by-Step: Lecture 3 Example 6**

The problem in this SolidWorks simulations demo is from Lecture 3 Example 6 in the Equilibrium lecture:

Graphical user interface, application

Description automatically generated

In this problem, we can apply the 4kN load to a 3D approximation of the above model in SolidWorks. Using SolidWorks, we can confirm and compare with the solutions derived in lecture.

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.

1. Download the L3E6 folder and unzip the contents (Or open the file using Citrix). You should see a model as below.

Graphical user interface, application, Word

Description automatically generated

1. Make sure that the Simulations tab is visible in your SolidWorks window. Right click the tool bar 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 will allow you to define component interactions, connections, external loads, and to generate a mesh.

Graphical user interface, text, application

Description automatically generated

Graphical 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 both supports on the beam and the support beam as below.

Graphical user interface, application

Description automatically generated

Click the checkmark to confirm your fixtures.

We have two options for applying a force at the end of the beam. (1) The first is to add a distributed load on the surface at the end using a sketch overlayed on the beam or (2) to use reference geometries to apply a remote load in the center of the end.

1. Method 1: use the dropdown arrow in the External Loads Advisor and select the Force/Torque option. Then select the surface at the end to apply a distributed load. Change the force value to 4000 N.

Graphical user interface, application

Description automatically generated

Enter 4000 N at the bottom of the panel. For the second option, go ahead and delete the force by clicking the X instead of the checkmark.

1. Method 2: The second method requires going into the Assembly tab, clicking the dropdown menu for Reference Geometry, click on the corner point of the beam as below. In the x-axis box, click the edge of the top of the I width and in the z-axis box, click the length edge of the beam as indicated below. The red arrow is the x-axis and the blue arrow is the z-axis.

Graphical user interface, application

Description automatically generated

Click the checkmark to confirm the coordinate system.

Graphical user interface

Description automatically generated with low confidence

Switch back to the Simulations tab.

1. Method 2: Click the External Loads Advisor dropdown menu and select Remote Load/Mass.

Graphical user interface

Description automatically generated

1. Method 2: This should open a side panel as below. In the face selection section, select both faces encompassing the entire top of the beam. Then, for the reference coordinate system box, change the setting from Global to User defined. It should automatically select the coordinate system you’ve just made.

Graphical user interface

Description automatically generated

1. Method 2: To determine the length of the edges such that you can apply the remote load in the center of the beam, we can use the measure tool in the Evaluate tab. Click on Measure in the Evaluate tab with the Remote Loads/Mass menu still open.

Graphical user interface, application

Description automatically generated

1. Method 2: Select the corner where you placed the coordinate system and at the point of the split surface to determine the length as below, which is 0.66 ft dZ. Therefore, we want to place our point at 0.33 ft.

Graphical user interface

Description automatically generated with medium confidence

1. Method 2: Measure the width of the I-beam by clicking the edge. Here we measure it to be 0.49 ft, so we want our force to be at 0.25 ft for the x-coordinate.

Graphical user interface, application

Description automatically generated

1. Method 2: Enter 0.33 ft for z and 0.24 for x, respectively. Then, change the Y force to 4000 N, ensuring that the direction is correct using the switch direction button to the left of the location where you enter the force value as below. Ensure that your units are in N.

Graphical user interface

Description automatically generated

Click the checkmark afterwards to confirm your changes. You can also exit out of the measuring tool.

1. We need to apply a material to the object as well. We will use Plain Carbon Steel for the object, so right click the yellow part icon and go to Apply Favorite Material to All, then click Plain Carbon Steel.

Graphical user interface

Description automatically generated

1. Go to the Simulation tab and click the dropdown arrow on the Run This Study icon. Click on Create Mesh and click the checkmark to go with the default.

Graphical user interface, application

Description automatically generated

1. Now, we can run our study. Click Run This Study.

Graphical user interface, application, Word

Description automatically generated

1. In order to analyze our results, click on the Results Advisor dropdown menu and click List Result Force as below.

Graphical user interface, text, application

Description automatically generated

1. In the panel that opens for the List Result Force option, select the Reaction Force option and select the Right plane for the plane selection section in pink.

Graphical user interface

Description automatically generated

Use the dropdown of the L3E6 object icon as above to select the Right plane. Make sure units are SI.

1. Select the bottom of both supports, the same location where we fixed our supports.

Graphical user interface, application

Description automatically generated

1. Click Update to collect the reaction forces.

Graphical user interface, application

Description automatically generated

We can see that the FX (Ax) and FY (Ay) forces are similar to those calculated in the problem. Furthermore, Force at member BC is also equal to that of our calculations, confirming our methods.

Diagram

Description automatically generated