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

The problem in this SolidWorks simulations demo is from Homework 7 Problem 5, which asks you to determine the displacement due to an applied force on a system of cantilevered and simply supported beams as below.

Diagram

Description automatically generated

Using SolidWorks Simulations, we can also apply this load to a 3D approximation of this model to determine the displacement of the center of the beam.

1. Download the HW7P5 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, email

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 I cross section of the bottom I-beam.

Graphical user interface, application

Description automatically generated

Next, select the circular bottom of the support on the bottom of the simply supported top beam and click the checkmark to confirm your fixtures.

Graphical user interface

Description automatically generated

1. Next, we need to create the 15 kip load at the center of the beam. In order to accomplish this, we first need to create a reference geometry point to place our force. Select the dropdown arrow for the Reference Geometry icon and select Point, which will open a panel to the left.

Graphical user interface, text, application, Word

Description automatically generated

1. In the panel that opens, select the top face of the simply supported top beam and select the Center of Face option to create a reference point in the center of the face.

Graphical user interface, application, Word

Description automatically generated

Click the checkmark to confirm your selections.

1. Next click on the Force dropdown menu and select the Force option.

Graphical user interface, text, application

Description automatically generated

1. In the blue box, select the center point and change the directionality from Normal to Selected direction.

Graphical user interface, application

Description automatically generated

In the Selected direction options, choose the top face where you placed the reference point.

1. Change the units to English (IPS) and change the force values of the bottom box to 15000 lbf, which is equivalent to 15 kip. If the arrow is facing upwards, click reverse direction.

Graphical user interface, text, application

Description automatically generated

Click the checkmark to confirm your force.

1. Now, we need to apply a material to the object as well. We will use A36 steel for this system of objects. Right click the Parts icon and select Apply Material to All.

Graphical user interface, application

Description automatically generated

1. In the pop-up menu, search for A36 Steel and select ASTM A36 Steel.

Graphical user interface, application

Description automatically generated

Click Apply and close the menu.

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, text, application

Description automatically generated

1. We need to slightly change the settings for solving our system. Click the dropdown arrow for New Study and select Study Properties.

Graphical user interface, text, application

Description automatically generated

1. In the pop-up menu, change the solver from Automatic to Manual and select Intel Direct Sparse, which is necessary due to the morphology of the system.

Graphical user interface, text, application

Description automatically generated

1. Before running the Simulation, we need to alter the nature of the interactions between objects slightly. Double click the Component Interactions icon under the Connections icon in the left-hand menu.

Graphical user interface, text, application

Description automatically generated

1. In the menu for Component Interactions, under Interaction Type, select **NO PENETRATION.** This will differ from the below image due to different versions of SolidWorks, you should have the most recent version downloaded.

Graphical user interface, text, application, chat or text message

Description automatically generated

1. Because we changed the interactions, we need to recreate our mesh. 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. 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.

1. To analyze our results for displacement, right click the Displacement tab under the Results and select Probe. We can probe for results along the beam.

Graphical user interface, application

Description automatically generated

1. In the Probe Result menu, selection On selected entities in the Options section. In the blue box, select the edge of the top simply supported beam. Then, click update to collect the displacement values along the beam’s edge.

Graphical user interface, text, application

Description automatically generated

1. Scroll down to the Report Options and select Plot to graphically represent the results, you can also extract the values in an excel spreadsheet as well.

Graphical user interface, application

Description automatically generated

1. From the output graph, we can collect the values at a parametric distance of 0.5 (the center of the beam). Here, we see that the displacement is about 3 in, which is similar to the 3.23 in calculated using the superposition principle. Your graph will look slightly different due to the images here being derived from an earlier version.

Chart, scatter chart

Description automatically generated