**ME 371 – Mechanical Design II**
Lab Exercise #4

Part 1
Assembly of two components

In this part of the lab, two Lego beams with a rigid (welded) connection are simulated (Fig. 1a). This is the simplest way of connecting two parts in Fusion 360. For that, the basic steps will be as follows:

Step 1: Import and assemble the structures:

1. Download the parts (named "beam_9.prt"and "beam_3.prt") and save them in an appropriate directory. Then, create a new project folder (Figure 1(a)) and upload the two part-files **individually** (Figure 1(b)). If you upload both the files as a single upload, just keep in mind, one part will be saved under another part. After successful uploading, both beams should be found like the inset of Figure 1(b).
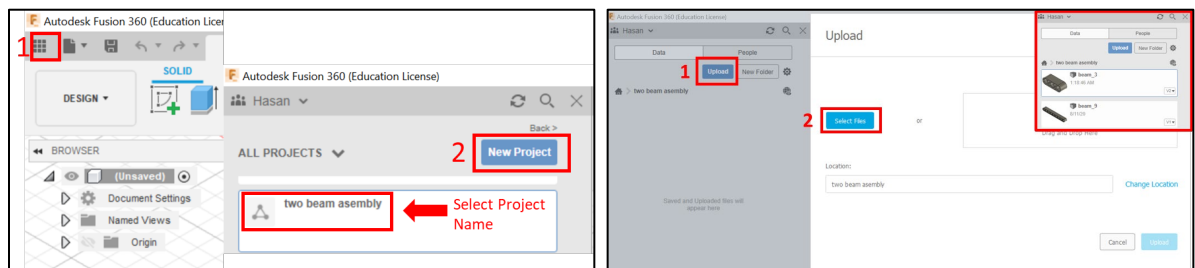
Figure 1 (a)Create a new project for assembly. (b) Upload parts in the project (inset: uploaded parts in data panel).

2. Now, save (*file>save*) the current design window first before starting the assembly operation. [if you do not save it, an error message will pop-up] Then, drag and drop one (say, beam_9) component after another one (say, beam_3) in the design workspace. The 2nd component may appear inside the first component (like figure 2(a)), which may create difficulties to find the individual component. To place components in suitable place, use the arrows (see the 3 orthogonal white arrows in Figure 2(a)) to place two parts in any arbitrary different locations. As both the beams are free to move, one beam needs to be grounded first (Figure 2(b)). To do this, follow the instructions as shown in Figure 2(b). Press *Beam_9*(right click)*>Ground*
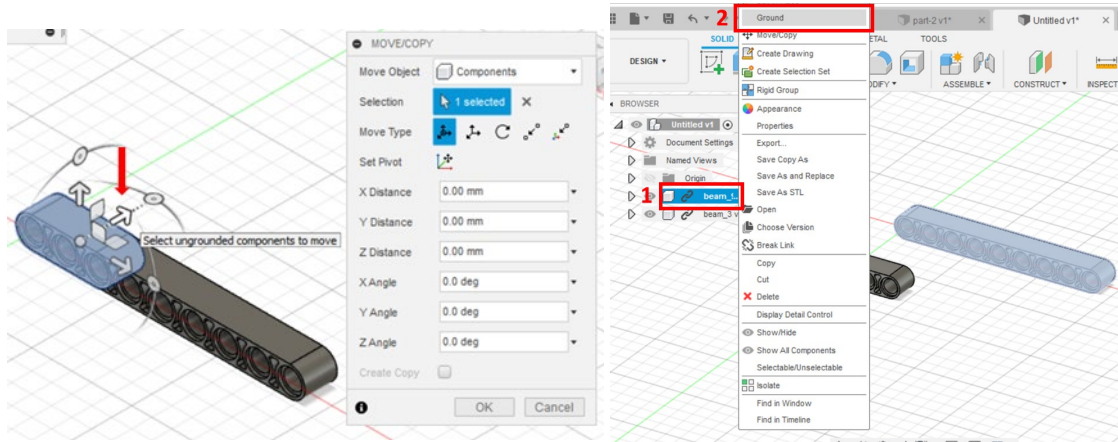
Figure 2: (a) Inserting components in design workspace by simply drag and drop. Red arrow shows the 3 orthogonal axes, which can be used to move the component. (b) Selecting the longer beam as ground by _Beam_9_(right click)_>Ground_.

3. To create a **rigid (welded) joint** between the components at particular location, use _Assemble>joint_ (or, press 'j' for shortcut). As illustrated in Figure 3(a), select the center point of the outer circle in the shorter beam as the snap of component one. [**Tip**: move the cursor until outer circle is highlited, then hold _control_ and select the center point of that circle]. Similarly, select the center of the outer rightmost circle as the snap of component two (Figure 3(b)). If everything is selected perfectly, the joint should be look like Figure 4.

→ You can explore the options in the "Motion" tab of the Joint menu for different types of connections. "Rigid" is the default. We'll choose another connection type later in this lab.
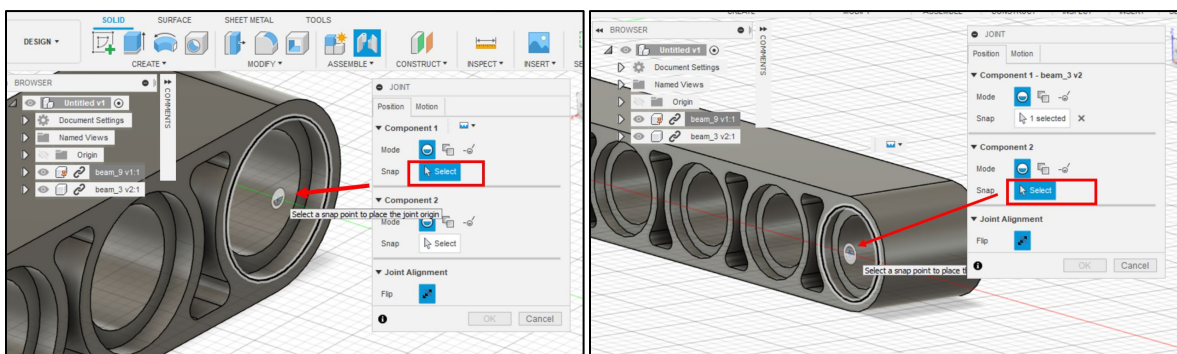
Figure 3 For welded fixed joint, select (a) center point of shorter beam and (b) center point of longer beam as two components.
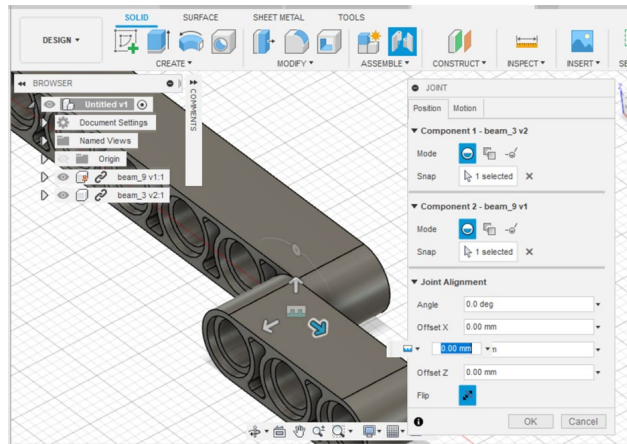
Figure 4: If points are selected properly for creating fixed joint in assembly, two beams should be placed like above, then press Ok.

4.   Now, open the simulation module from _design>simulation_ (if you have forgotten how to do this, review the previous Lab) and create a new 'static stress' study.

Step-2: Assign material

1.   Assign **PC/ABS Plastic** as the beam material for both beams. For that, select _Study Materials>(Same as Model)>_ PC/ABS Plastic.

Step-3: Apply constraints and loads

1.   Applying **Fixed** constraints on the left most cylinder of the longer beam using _constraints (right click)>structural constraints_ (Figure 5)

2.   Apply **5N force along X-axis on both edges** of the short beam as shown in Figure 5. For selecting both edges, select one edge, hold _shift+ scroll_ key in mouse to rotate for suitable view, hold the _shift_ key again and select another edge (Figure 5).
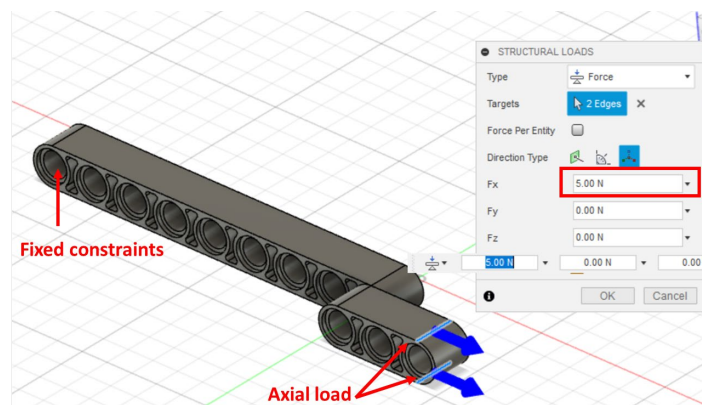
Figure 5: Apply fixed constraints at longer beam. Then, apply 5N axial force on both edges of the shorter beam

Step-4: Apply mesh and precheck

1.   For generating automatic mesh, select _Mesh>Generate Mesh_.

2.   Then, perform a Pre-check (_Results>Pre-check_) to make sure all materials, constraints, loads, and mesh settings are properly assigned.

Step-5: Run simulation and analyze results

1.   To run the simulation, use _Results>Solve_ (just below the pre-check option) from the left browser tree.

2.   After simulation, view the results to examine the contour plot of stress, reaction force, contact pressure and displacement.

> **Tip**: To more easily view the stresses in the individual components, you can toggle the component visibility. You can do this under the browser tree -> model components -> click the "eye" symbol to toggle visibility on/off for each component

Part 2
Two Lego beams welded to a connector

Step 1: Import and assemble the structures

1.   Download and import the parts (Use **beam_9**.prt, **beam_connect_1**.prt, **beam_3**.prt) similar to Part-1 of this lab manual. During assembly, you may need to **rotate** the part named beam_connect_1.prt by **90⁰** along X-axis (Figure 6(a)). Unlike one welded fixed joint between two beams in the previous section, this time, we will make two welded fixed joints connecting the three parts as shown by the red arrows in Figure 6(b). To do this, follow the same process of making fixed joint between the components mentioned in previous section.

2.   Just like previous section, open the simulation module from Design drop-down menu and create a new 'static stress' study.
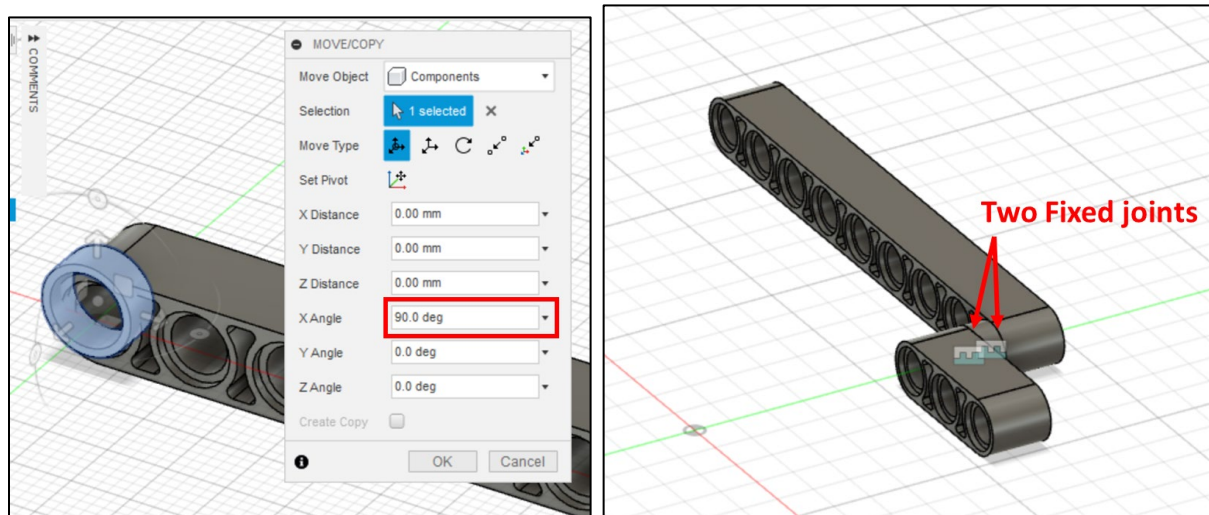

Figure 6 (a) Rotate the part named beam_connect_1.prt by **90⁰** (if necessary) (b) Apply two fixed (rigid) joints.

Repeat Step-2 to Step-5 from Part 1 of this lab to apply constraints apply loads, generate the mesh and run the simulation. This time, we will not change any constraint or load. That means, **fixed constraint** at left most cylindrical surface of the longer beam, **axial line load of 5N** at both rightmost edges (top and bottom edges) of the shorter beam.

Part 3
Two Lego beams connected by a pin

Step-1: Import and assemble the structures

1.   Import three components named **beam_9**.prt, **conn_axle_blue**.prt, and **beam_3**.prt.
2.   Apply **rigid Joint** between the pin (conn_axle_blue.prt) and the longer beam (beam_9.prt). Then, apply **revolution joint** between pin and shorter beam (beam_3.prt). For revolution joint, you need to change motion type from rigid (by default) to revolution. Also, be careful on selecting the appropriate point from each component during this process. Final joint should be look like Figure 7. To check whether both the joints are made properly or not, you can hold the shorter beam and should be able to rotate it against the pin and longer beam assembly.

3.   Just like previous section, open Simulation module from Design drop-down menu and create a new 'static stress' study.
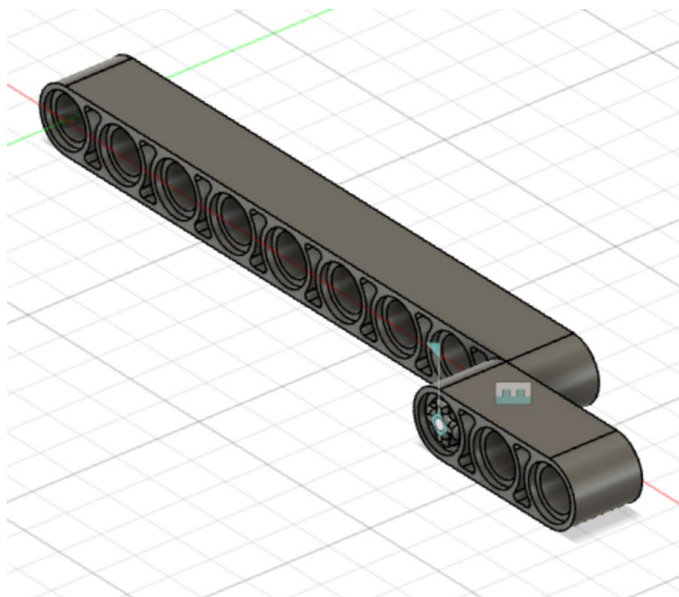

Figure 7 Two joints are made. **Rigid joint**: between pin and longer beam. **Revolve joint**: between pin and shorter beam.

Now, **follow Step-2 to Step-5 as mentioned in Part-1 of this lab** to apply loads and constraints and simulate the results.

Lab 4 questions

Question 1
Submit a screen shot that shows the von Mises stress distribution on the deformed assembly from the analysis conducted in **Part 1**. In addition, answer the following questions:
   a) What appears to be the primary mode of deformation of the assembly (axial stretching/compression, torsion, bending)? Be sure to plot the "Adjusted" deformation of the assembly to help answer this question.
   b) What is the maximum von Mises stress in **each** component? Use the "Tip" box on Page 4 of the tutorial to toggle the visibility of each part on/off in order to clearly identify the maximum von Mises stress in each part (no need to include additional screen shots for this step).
   c) Does the maximum von Mises stress occur at a location that you anticipated? Why or why not?

Question 2
Submit a screen shot that shows the von Mises stress distribution on the deformed assemblyfor the analysis conducted in **Part 2**. In addition, answer the following questions:
   a) What is the maximum von Mises stress in the assembly? In which part does it occur?
   b) How does the maximum von Mises stress compare to the model in Part 1? Offer an explanation for why the maximum value has changed in this assembly despite the magnitude of the applied load being the same.

Question 3
Submit a screen shot that shows the von Mises stress distribution on the deformed assembly from **Part 3**. In addition, answer the following questions:
   a) What is the maximum von Mises stress in the assembly? In which part does it occur?
   b) How does the location of the maximum von Mises stress change in this model compared to the models in Parts 1 and 2? Does the change in location coincide with your expectations? Why or why not?

Question 4
FEA Labs 3 and 4 represent two different modeling approaches:

   1) **Isolate a single component of interest** and apply loads that simulate not only the external loads that act on it but also its connections to other components or external supports, e.g. the "Bearing Load" used in FEA Lab 2 to simulate connection to a bearing which allows rotation (but not translation) at the connecting joint location.

   2) **Simulate an entire assembly** (two or more parts) subjected to external loads, where the transfer of loads between components depends on how the connections between the components are defined (what we did in this lab, e.g. rigid joint vs revolve joint).

List at least two specific situations or reasons that would favor both modeling Approach 1 and Approach 2. Consider that Approach 2 is in general more complex to set up and more costly (in terms of computer resources required to solve the model) compared to analyzing a single component as in Approach 1, so modeling an entire assembly should be done only when there is a clear reason to do so.