

Beam ASSEMBLY ANALYSIS

A. Activate Configuration.

Step 1. If necessary, open your **Beam Assembly** file.

Step 2. **Right click Beam Double** in the Feature Manager and click **Configure Component** from the Content toolbar, **Fig. 1**.

Step 3. In the Modify Configurations dialog box, click the **drop down arrow** of Configuration and select **BEAM COMBINE**, click **Apply** and **Close**, **Fig. 2**.

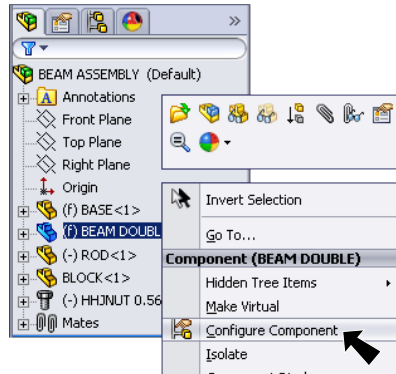


Fig. 1

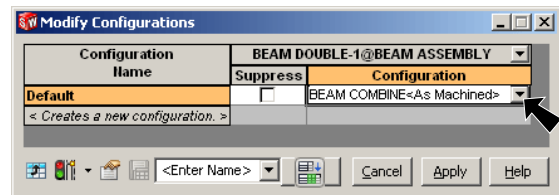
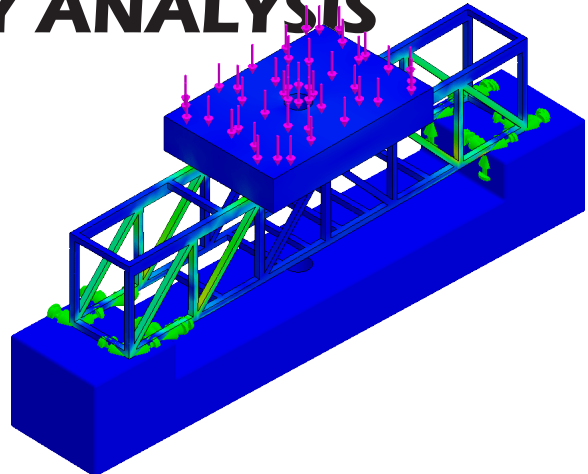



Fig. 2

B. Suppress Rod and Nut.

Step 1. **Suppress Rod and Nut Parts** in the Feature Manager, **Fig. 3**. This will speed up the solver time to run. To suppress the Parts, **ctrl click** the Rod and Nut Parts and click **Suppress**  from menu.

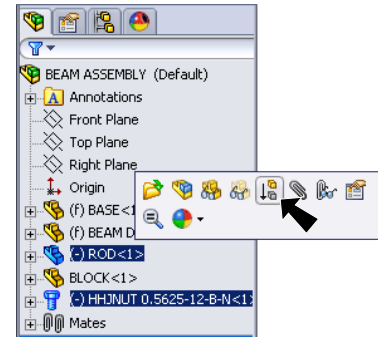
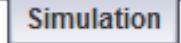



Fig. 3

C. New Study.

Step 1. Click **Simulation**  on the Command Manager toolbar.

Step 2. Click the **drop down**  of **Study Advisor**  on the Simulation toolbar and click **New Study**.

Step 3. In the Study Property Manager:
 under Name, **Fig. 4**
 key-in **Assembly**
 under Type
 select **Static**
 click OK .

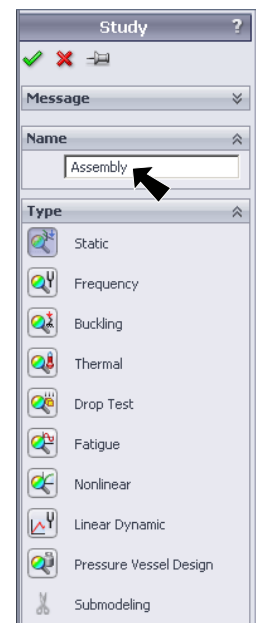


Fig. 4

D. Contact Sets.

Step 1. Click the **drop down**  of **Connections Advisor** on the Simulation toolbar and click **Contact Sets**.

Step 2. In the Contact Sets Property Manager:

under **Contact**, **Fig. 5**

select **Automatically find contact sets**

under **Components**

click **Beam Double** in graphics area, **Fig. 6**

check **Find contacts with the rest of the assembly**, **Fig. 5**

click **Find contact sets button**

under **Results**

Ctrl click each contact set, **Fig. 5** and **Fig. 7**

click **Create contact sets** 

click **OK** 

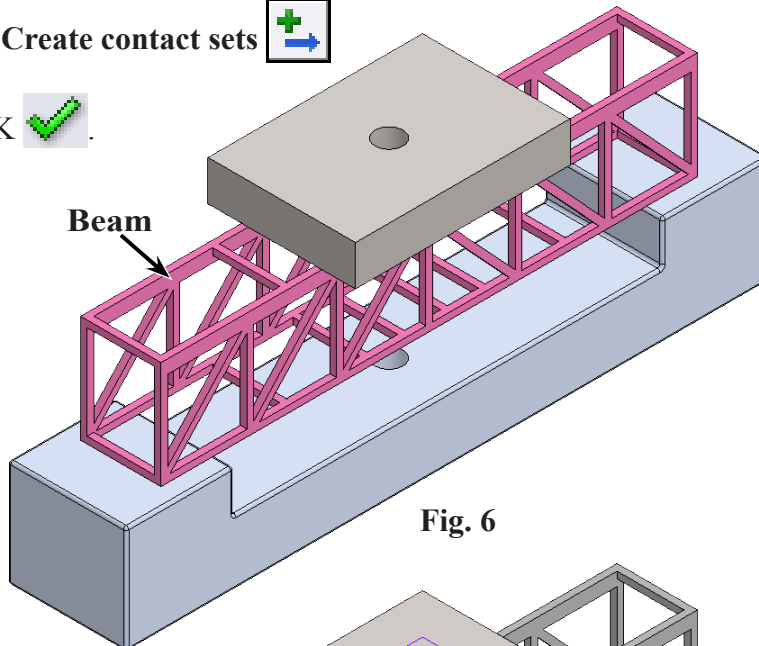


Fig. 6

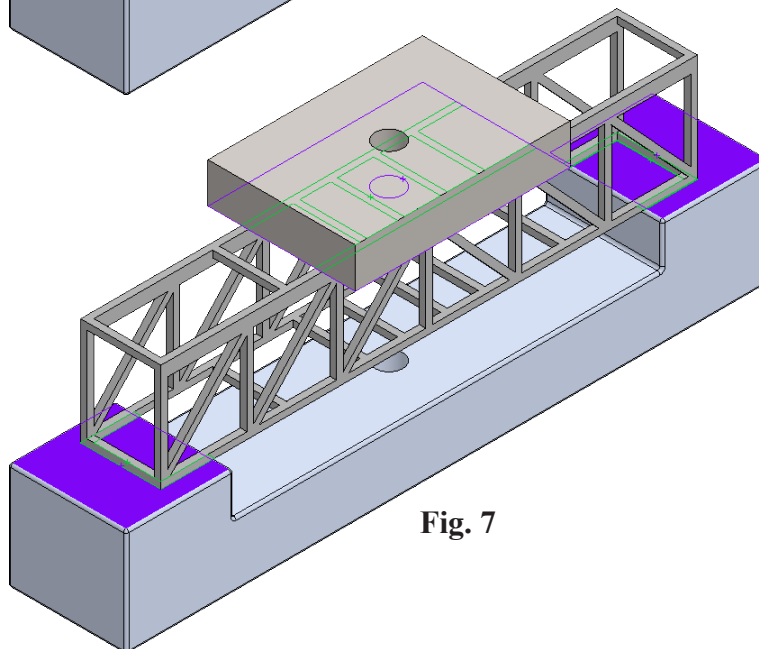


Fig. 7

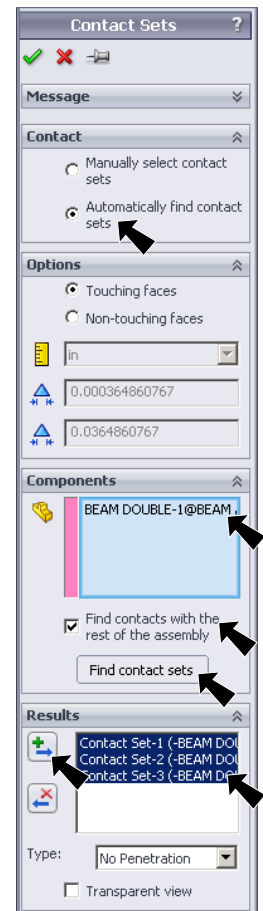
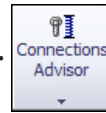



Fig. 5

E. Component Contact.

Step 1. Click the **drop down**  of **Connections Advisor** Simulation toolbar and click **Component Contact**.



on the

Step 2. In the Component Contact Property Manager:
under Contact Type, **Fig. 8**
select **No Penetration**
under Components
drag a selection around all Parts, **Fig. 9**
click OK .

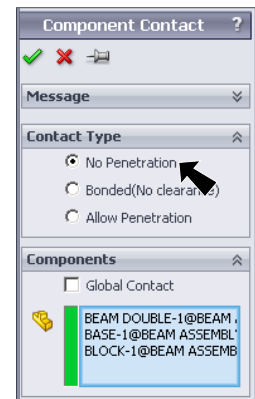


Fig. 8

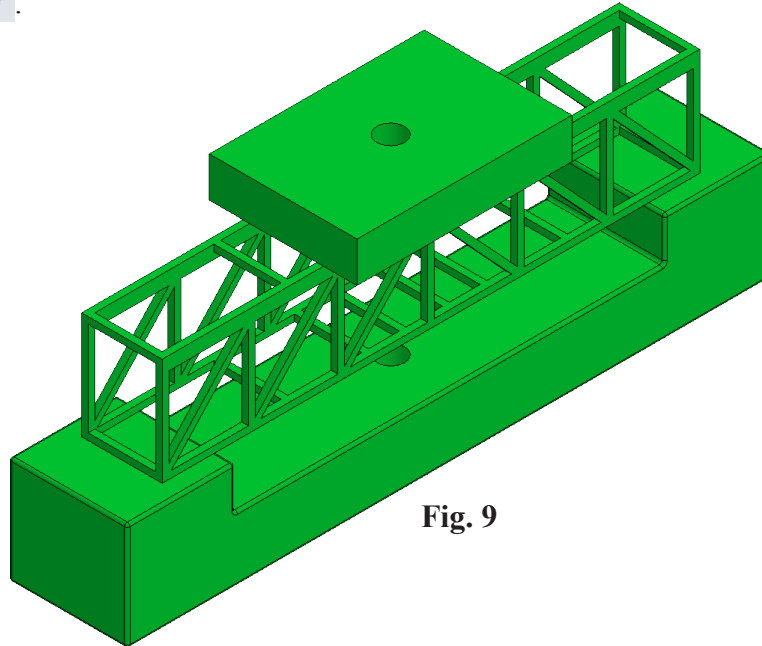


Fig. 9

F. Fixture.

Step 1. Click **Bottom**  on the Standard Views toolbar. (Ctrl-6)

Step 2. **Hide Base Part.** To hide, move cursor over the Base and press **Tab** key on keyboard, **Fig. 10**.

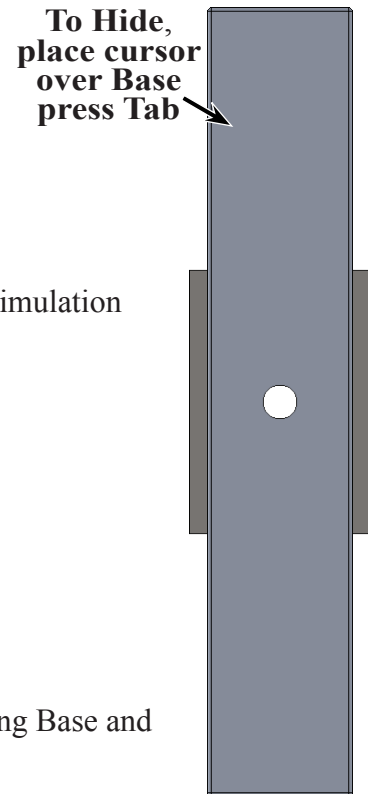




Fig. 10

Step 3. Click the **drop down**  of **Fixtures Advisor**  on the Simulation toolbar and click **Fixed Geometry**.

Step 4. In the Fixture Property Manager:
under Standard, **Fig. 11**

select **Fixed Geometry** 
in the Face, Edges, Vertices for Fixture field
click the **two bottom split faces** on Beam, **Fig. 12**
click OK .

Step 5. **Show Base Part.** To show, move cursor over the area containing Base and press **Shift-Tab** on keyboard, **Fig. 13**.

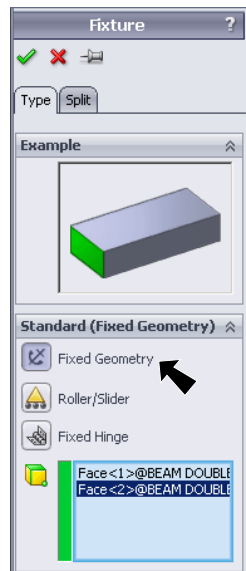


Fig. 11

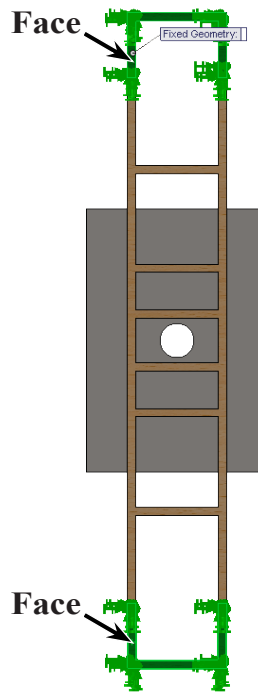


Fig. 12

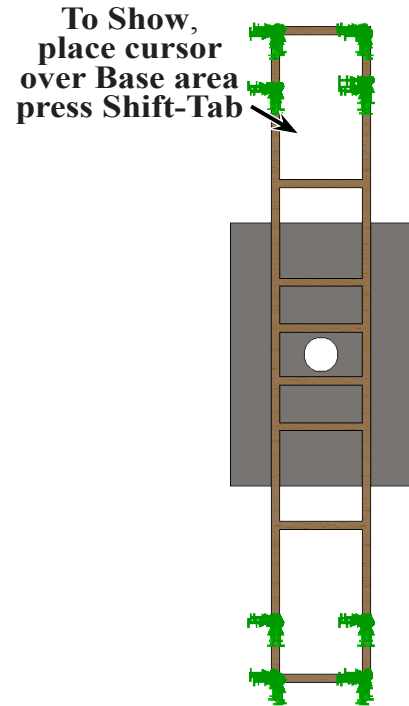


Fig. 13

G. Force.

Step 1. Click **Isometric**  on the Standard Views toolbar. (**Ctrl-7**)

Step 2. Click the **drop down**  of **External Loads...**  on the Simulation toolbar and click **Force**.

Step 3. In the Force/Torque Property Manager set:
under Selection, **Fig. 14**

select **Force** 

click **top face of Block**, **Fig. 15**

under Units

select **English**

set **Force Value 50**

click **OK** .

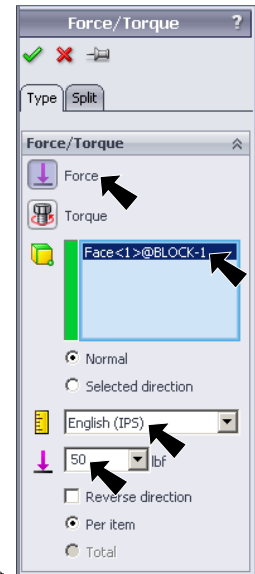


Fig. 14

Step 4. Save. Use **Ctrl-S**.

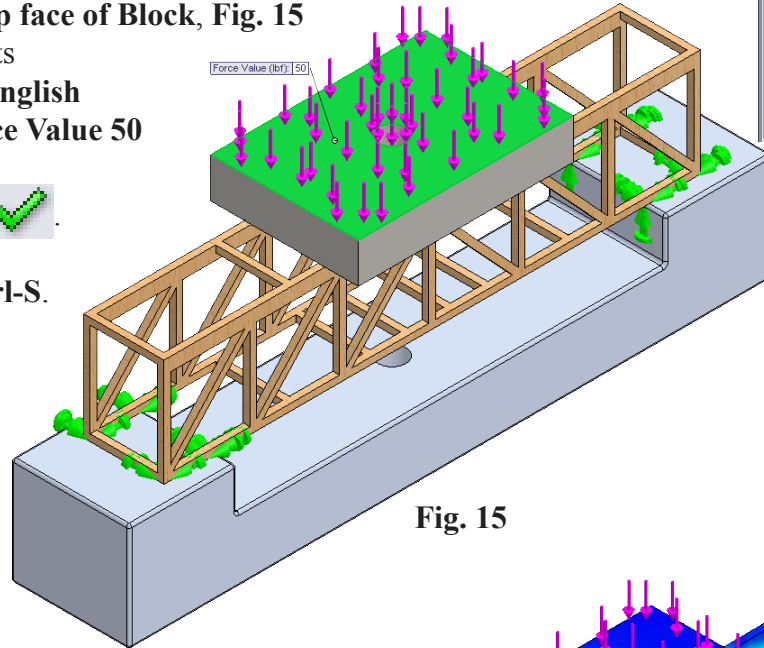


Fig. 15

H. Run.

Step 1. Click **Run**  on the Simulation toolbar.

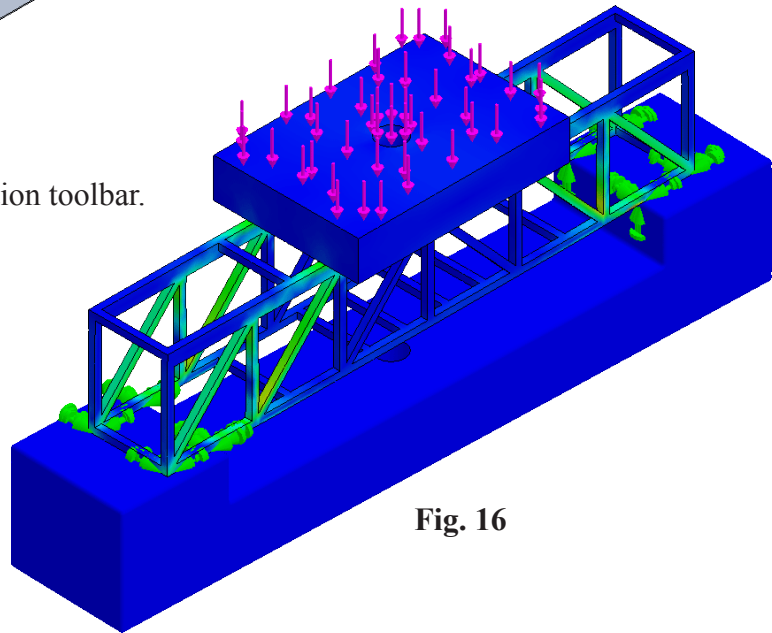


Fig. 16

I. Results - Factor of Safety.

Step 1. **Right click Results folder** in the Simulation Study Tree and click **Define Factor Of Safety Plot**, Fig. 17.

Step 2. In the Factor of Safety Property Manager, Fig. 18

Step 1 click Next 

Step 2 confirm **Multiplication factor is 1** and click Next 

Step 3. In the Factor of Safety Property Manager **Step 3**, Fig. 18

select **Areas below factor of safety**
note **Minimum factor of safety 1.31472**
your FOS might be slightly different

click OK .

Step 4. View the FOS plot. The red members will not support the current load. Blue members indicates we can increase the load.

Step 5. Next, we will change load using formula
current FOS * current Force = safe load
or $1.31742 * 50 \text{ lbs.} = 65.736 \text{ lbs.}$
Try 65.736 lbs of force.



Fig. 17

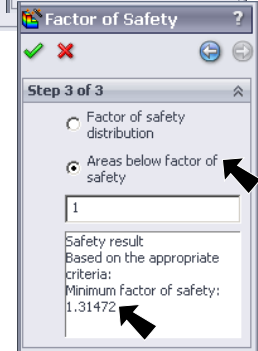


Fig. 18

J. Edit Force.

Step 1. **Double click Force1** in the Simulation Study Tree to display the Force Property Manager, Fig. 19.

Step 2. In the Force/Torque Property Manager set:
under Force, Fig. 20

Force Value 65.736

Tip: Key-in $*1.31742$ in the Force Value field after the 50, press Tab key and let SW do

the math 

click OK .

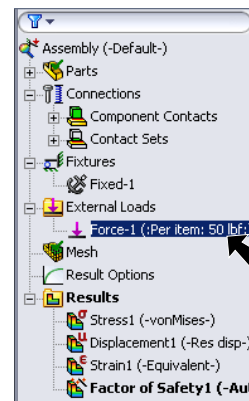


Fig. 19

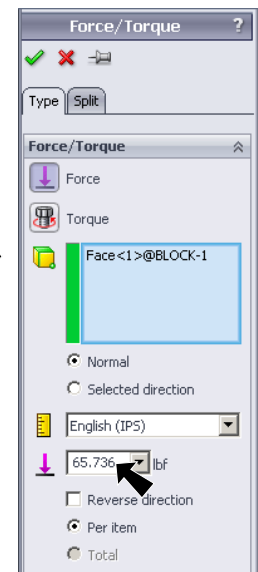


Fig. 20

K. Run.

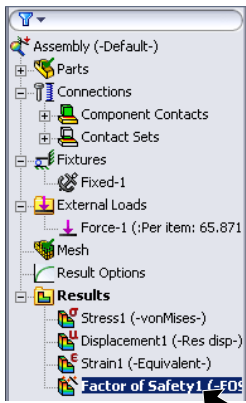
Step 1. Click **Run**  on the Simulation toolbar.

L. Confirm Factor of Safety = 1.



Step 1. **Double click Factor of Safety1** in the Simulation Study Tree to display the Factor of Safety1 plot, **Fig. 21**.

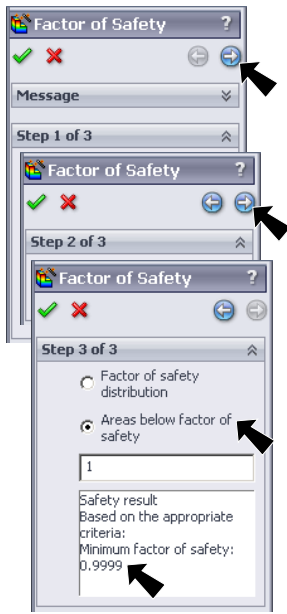
Step 2. All members should be blue. The blue member will support the load.


Step 3. **Double click Factor of Safety1** in the Simulation Study Tree to display the Force Property Manager, **Fig. 21**.



Step 4. In the Factor of Safety Property Manager, **Fig. 22**

Step 1 click **Next** 
Step 2 click **Next** 



Step 5. In the Factor of Safety Property Manager **Step 3**, **Fig. 22**
select **Areas below factor of safety**
note **Minimum factor of safety .9999**
your FOS might be slightly different
click OK .

Step 6. Save. Use **Ctrl-S**.