Welcome to

COSMOSWorks Tutorial

of

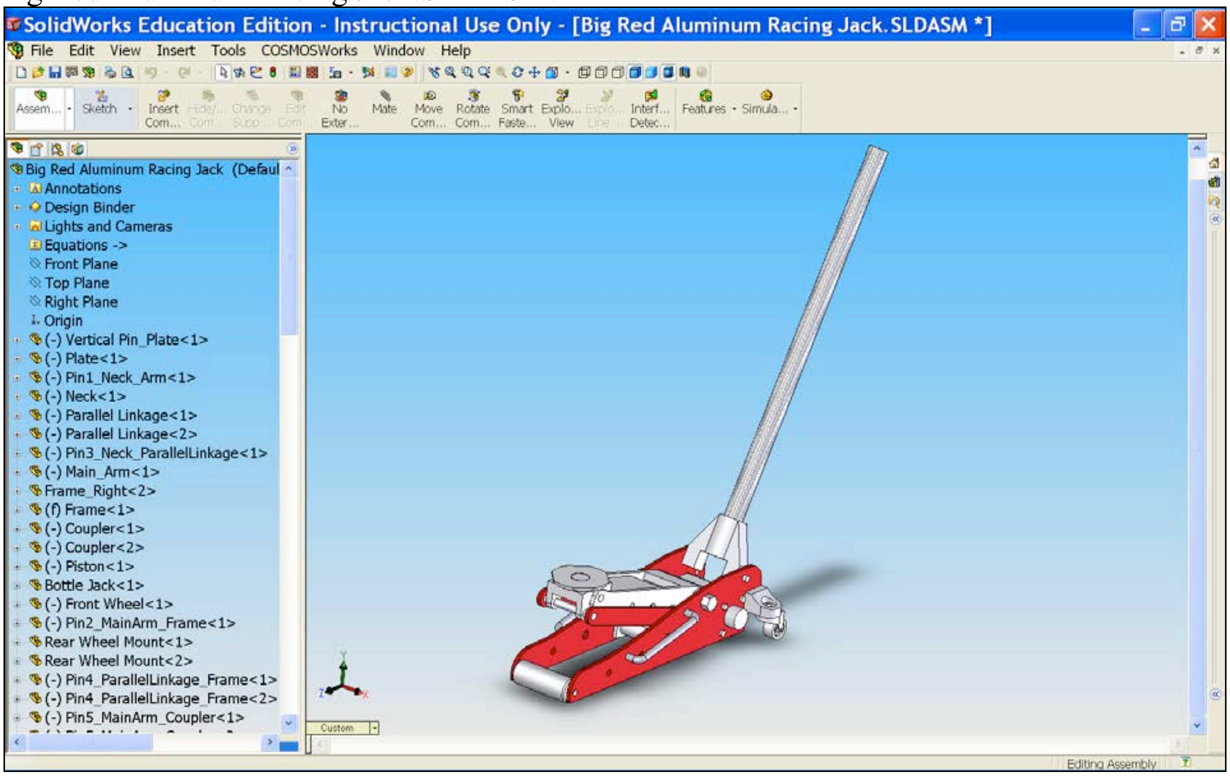
Torin BIG RED Aluminum Jack

Last updated by TJ, February 2013

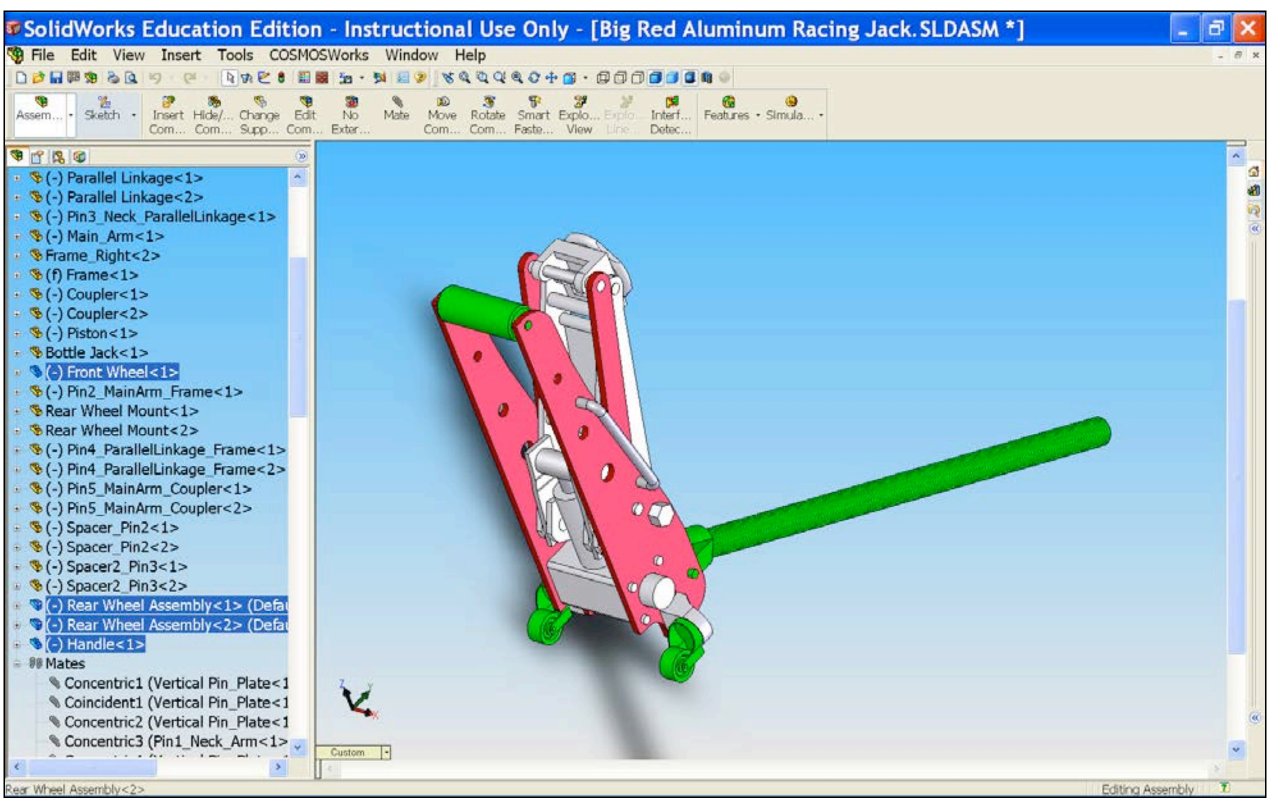
In this tutorial, we will use COSMOSWorks to perform stress analysis on an aluminum floor jack built by Torin. We will apply a maximum load that this floor jack can handle as claimed by the manufacturer and see if this floor jack can actually support that maximum load without failure on any of its component.

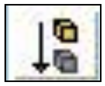
CosmosWorks Tutorial

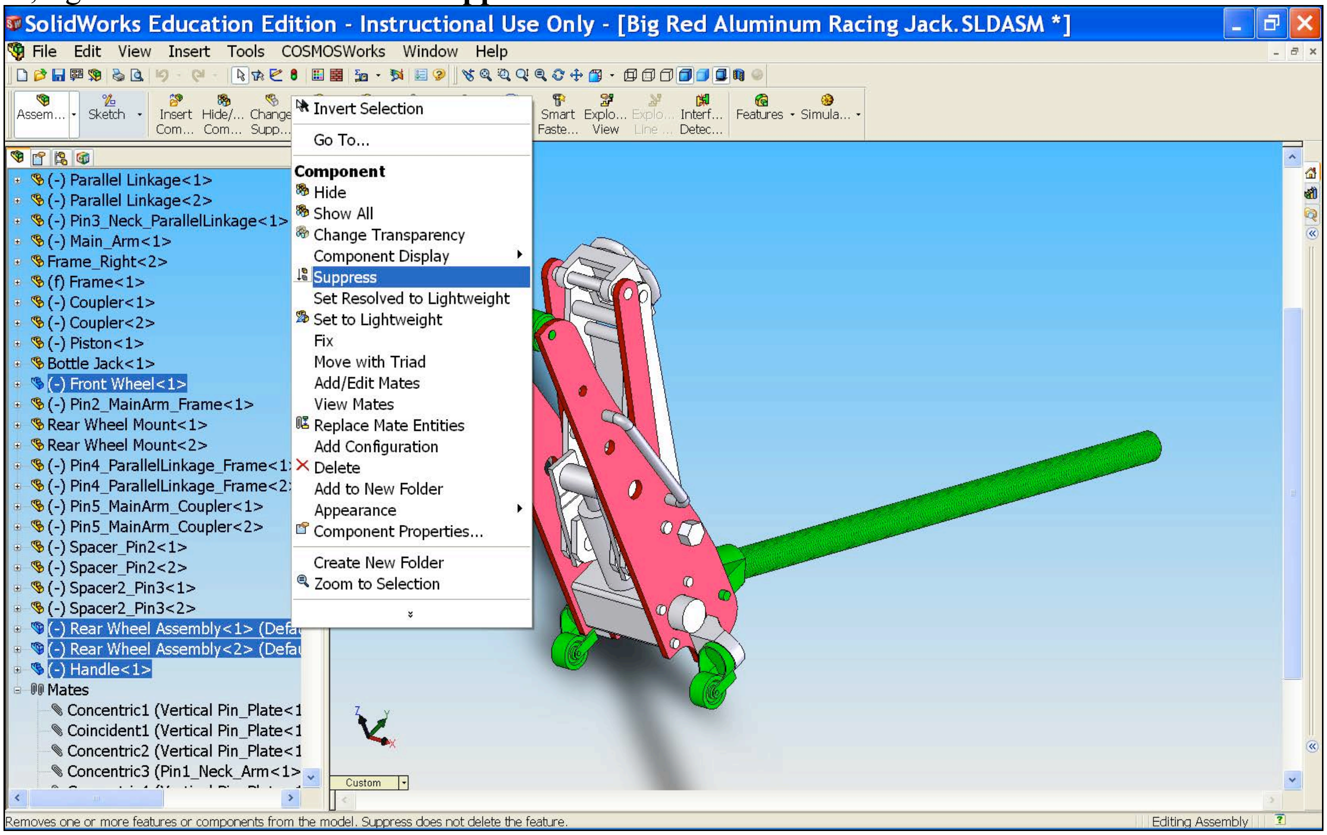
1) Open Big Red Aluminum Racing Jack.SLDASM.



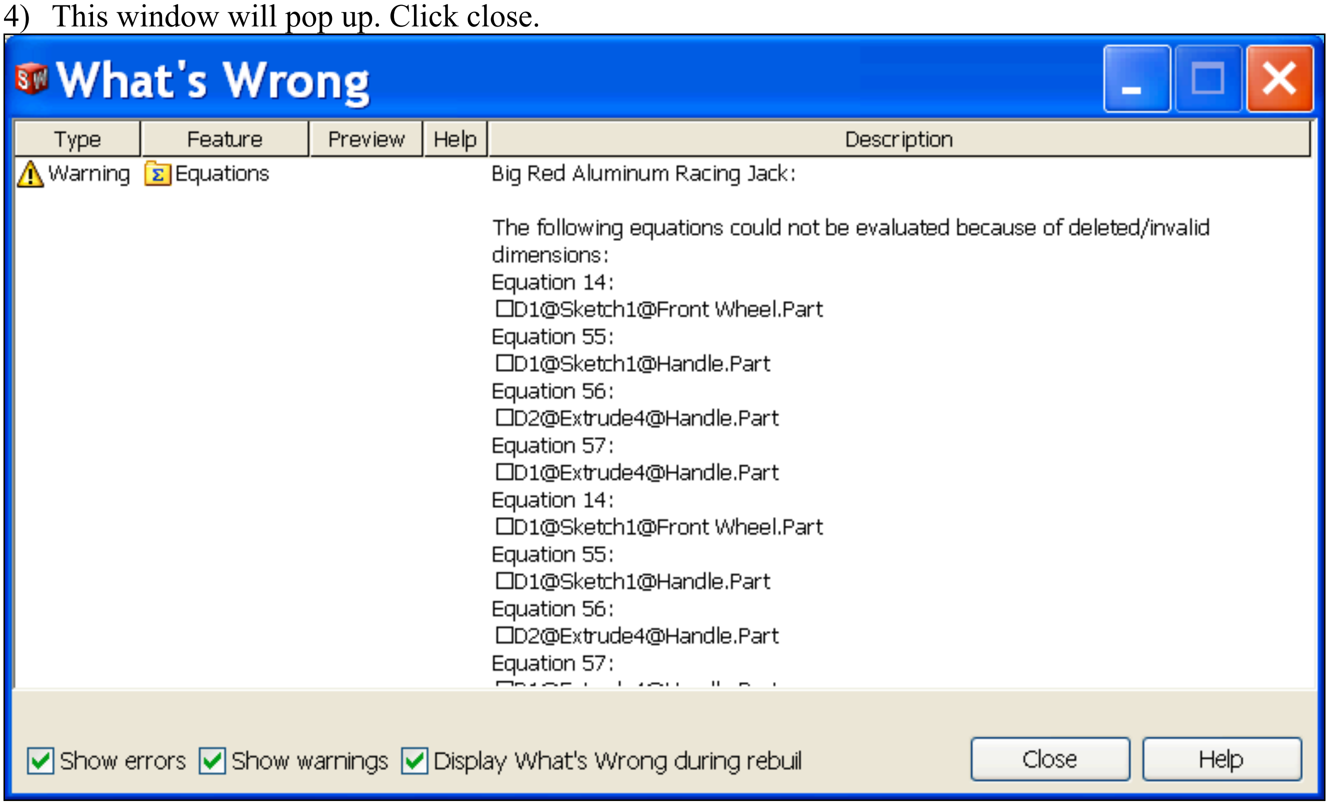
2) First of all, we need to suppress the front wheel, rear wheel assemblies and handle. To suppress all of those parts simultaneously, left-click **Front Wheel** and ctrl+left-click **Rear Wheel Assembly <1>**, **<2>** and **Handle**.



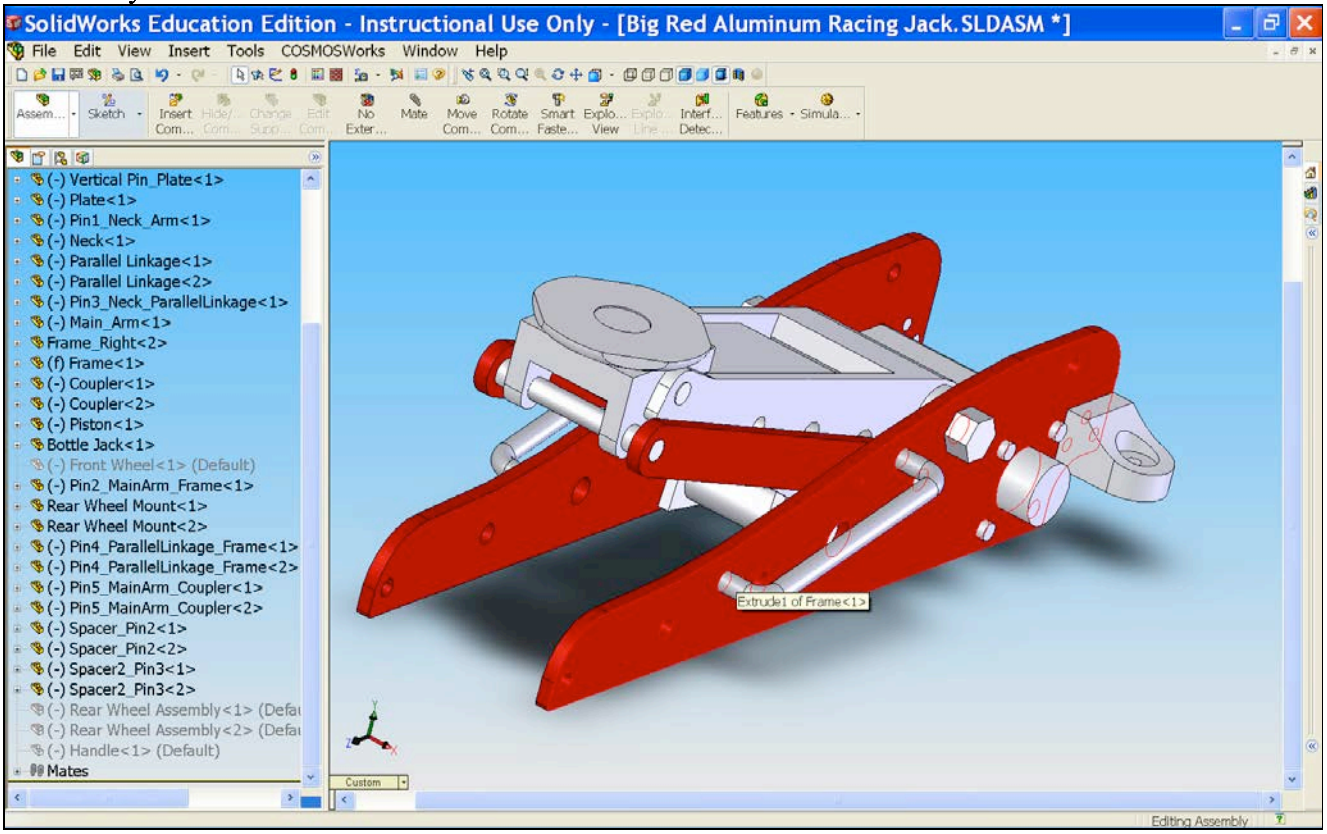
3) Now, right-click and select  **Suppress.**



4) This window will pop up. Click close.

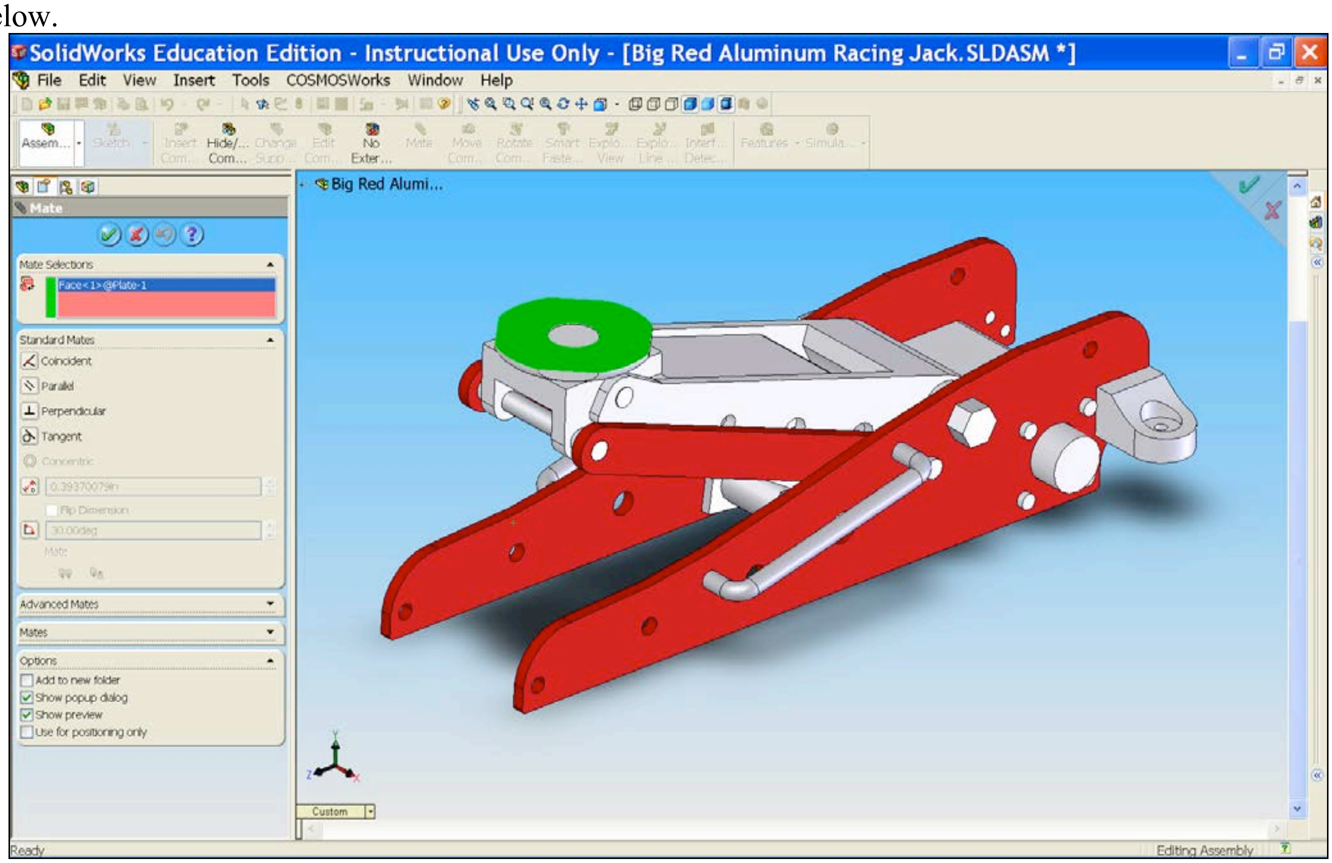


5) The assembly should now look like this.

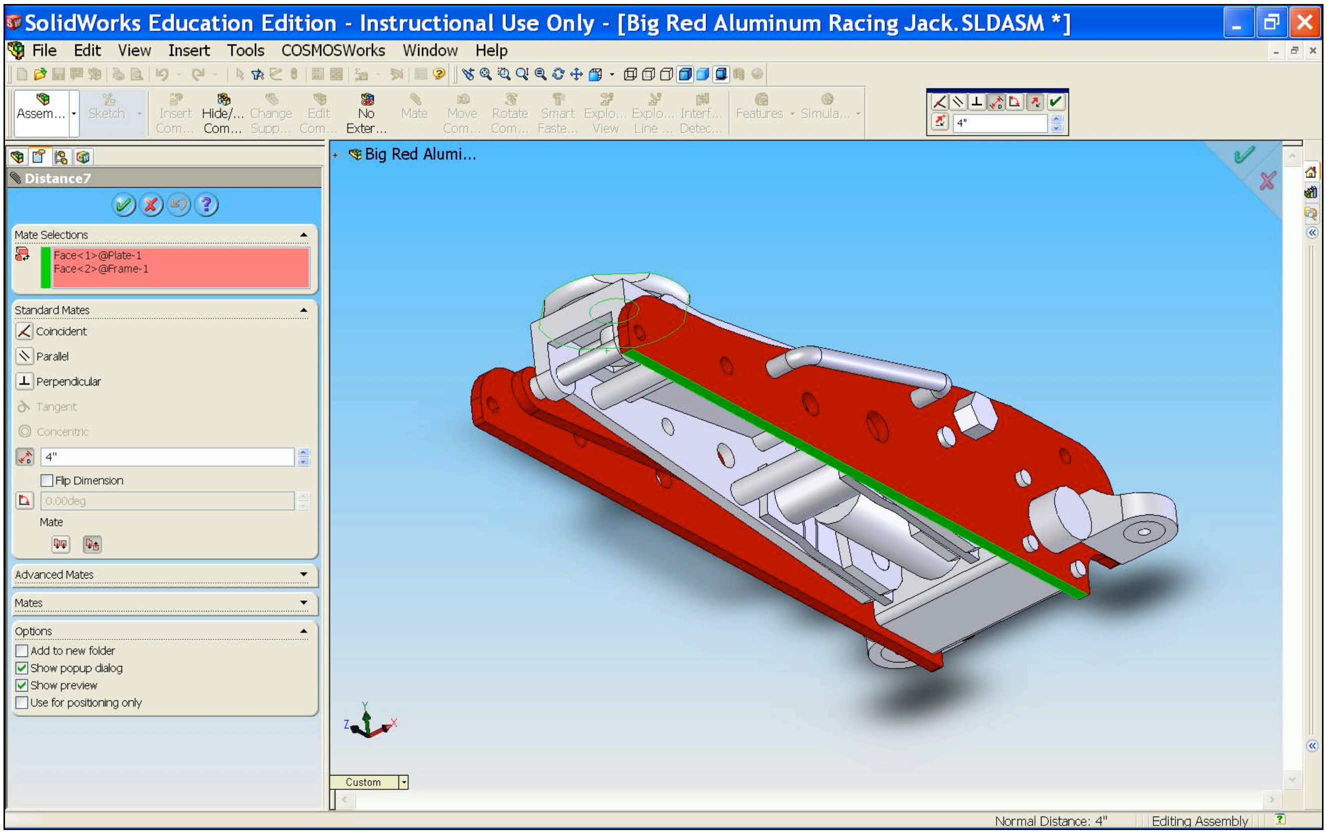


6) Now, we will define a distance between the plate, which supports the load from a car, and the ground. For this case, we assume the bottom surface of the frame as the ground. Click on 

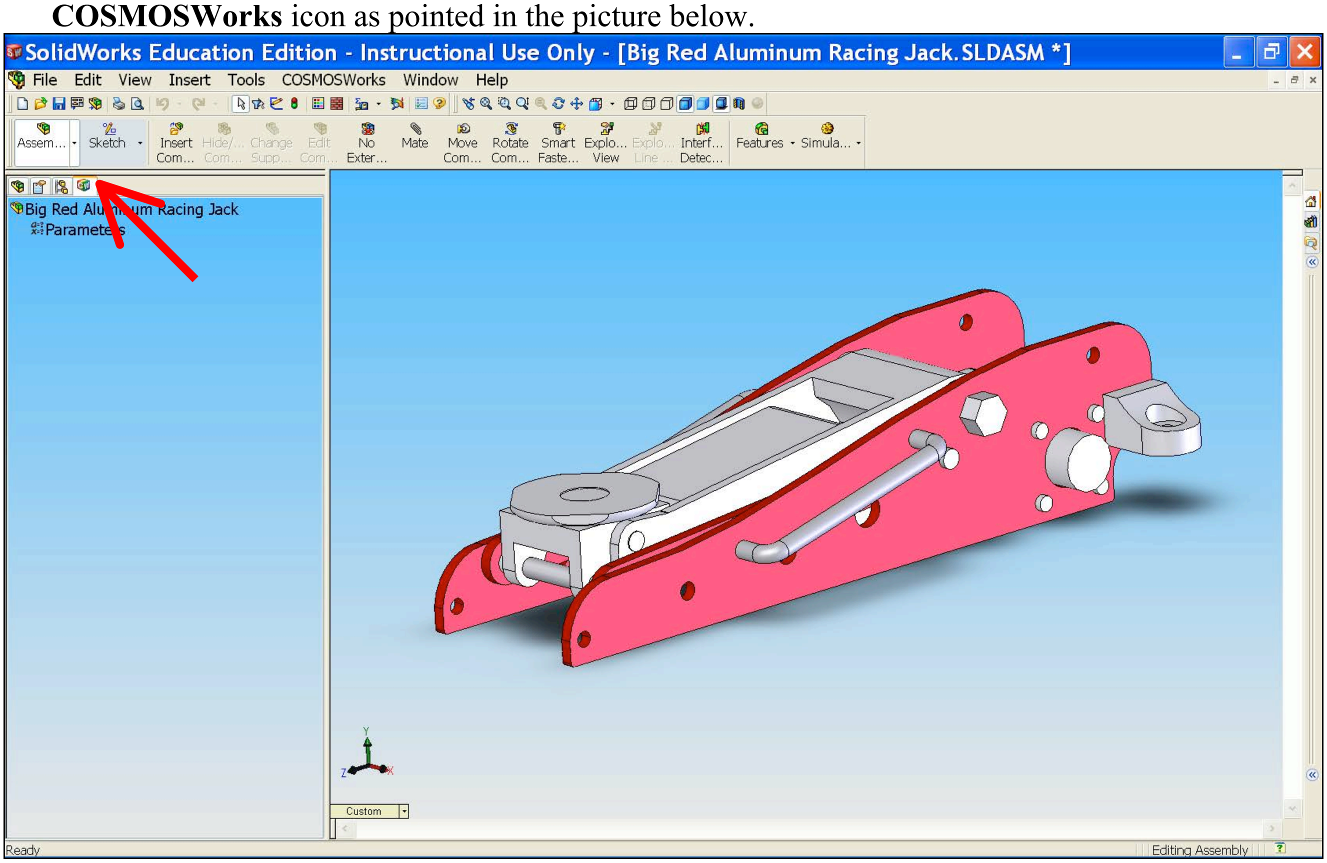
Mate. The Mate window will appear on the left. Under **Mate Selections**, we can first select top surface of the plate as shown in the picture below.



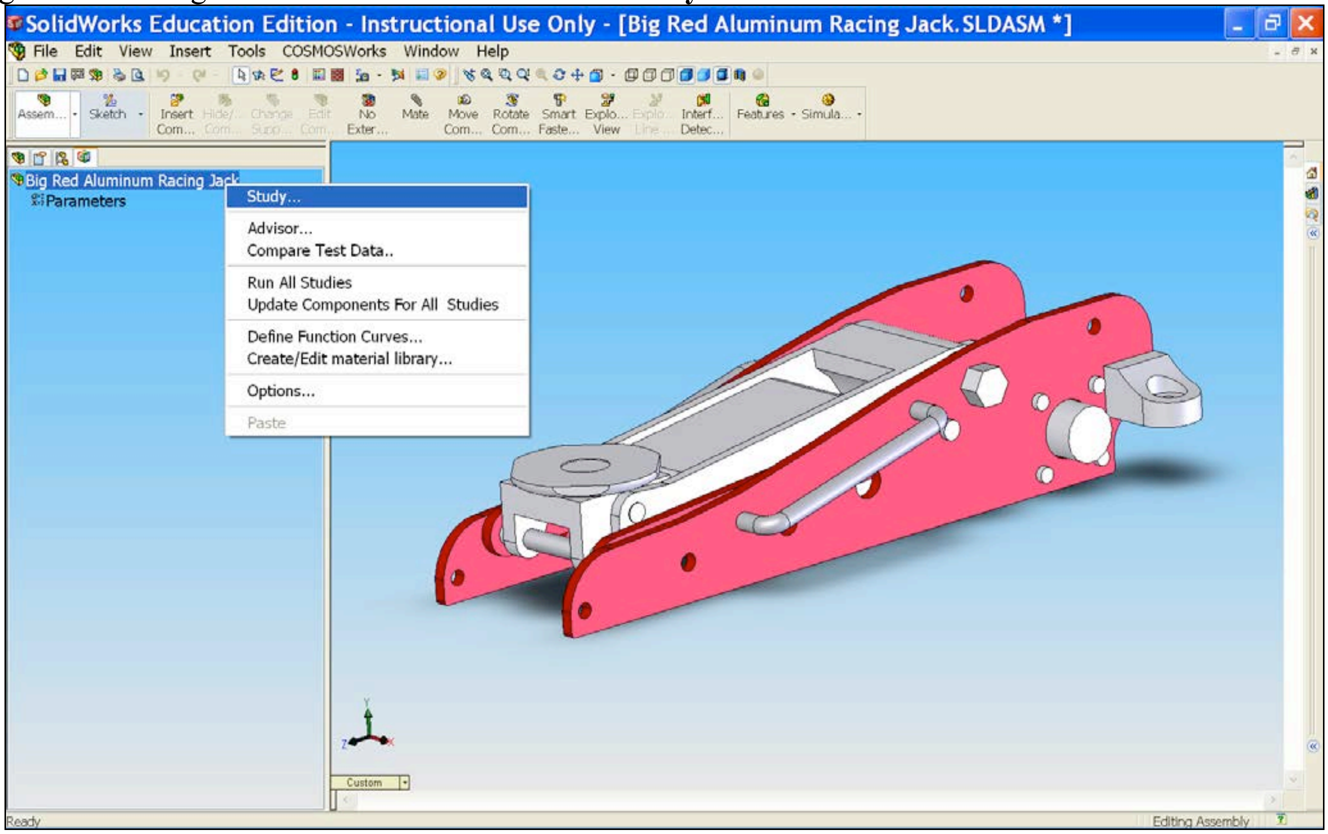
7) Second, select the bottom face of the frame as shown in the screen shot below. Under **Standard Mates**, click on  **Distance** and set the value to 4.21".



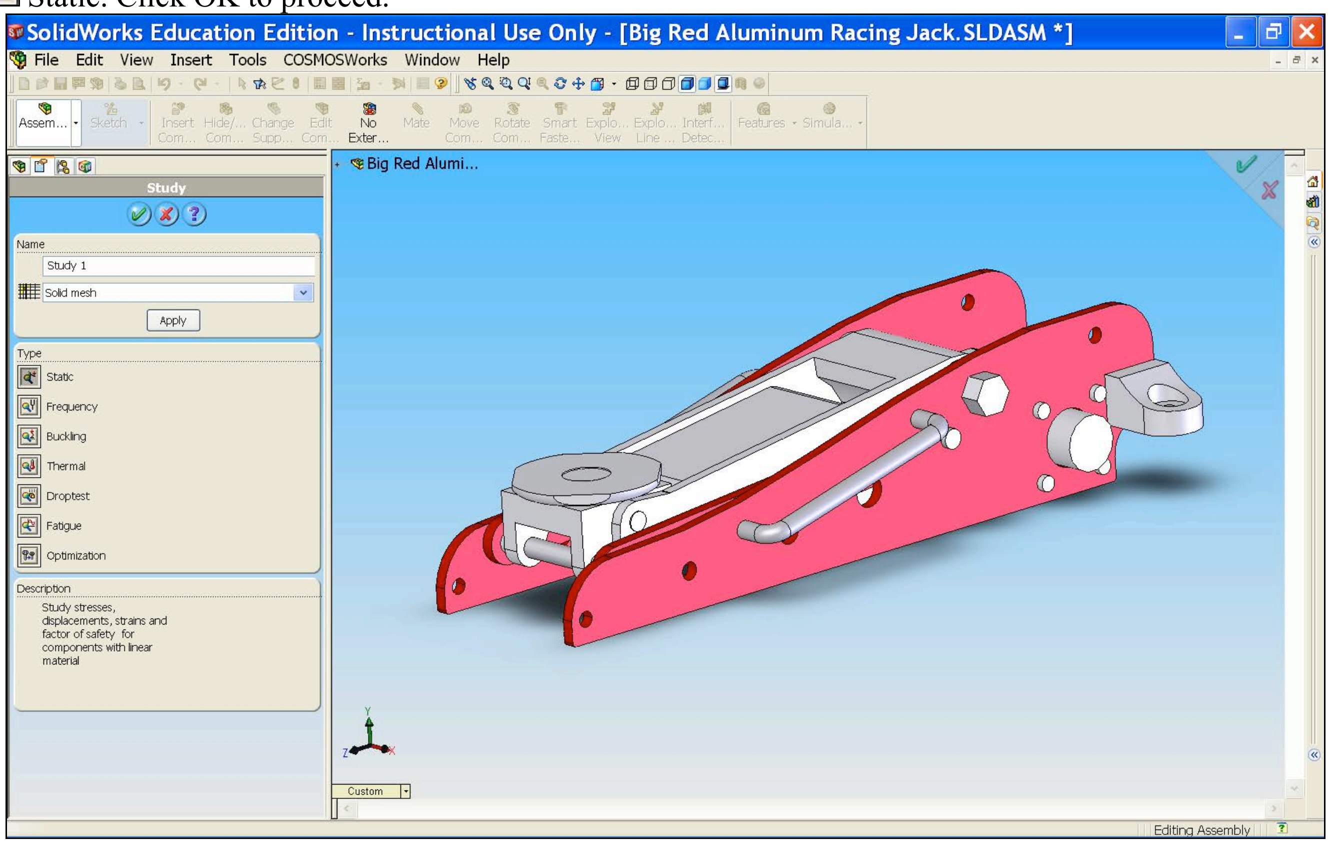
8) The top face of the plate and the bottom face of the frame are now 4.21" from each other. Now we will use COSMOSWorks to analyze the stress on the floor jack assembly when the load is applied. Click on the  **COSMOSWorks** icon as pointed in the picture below.



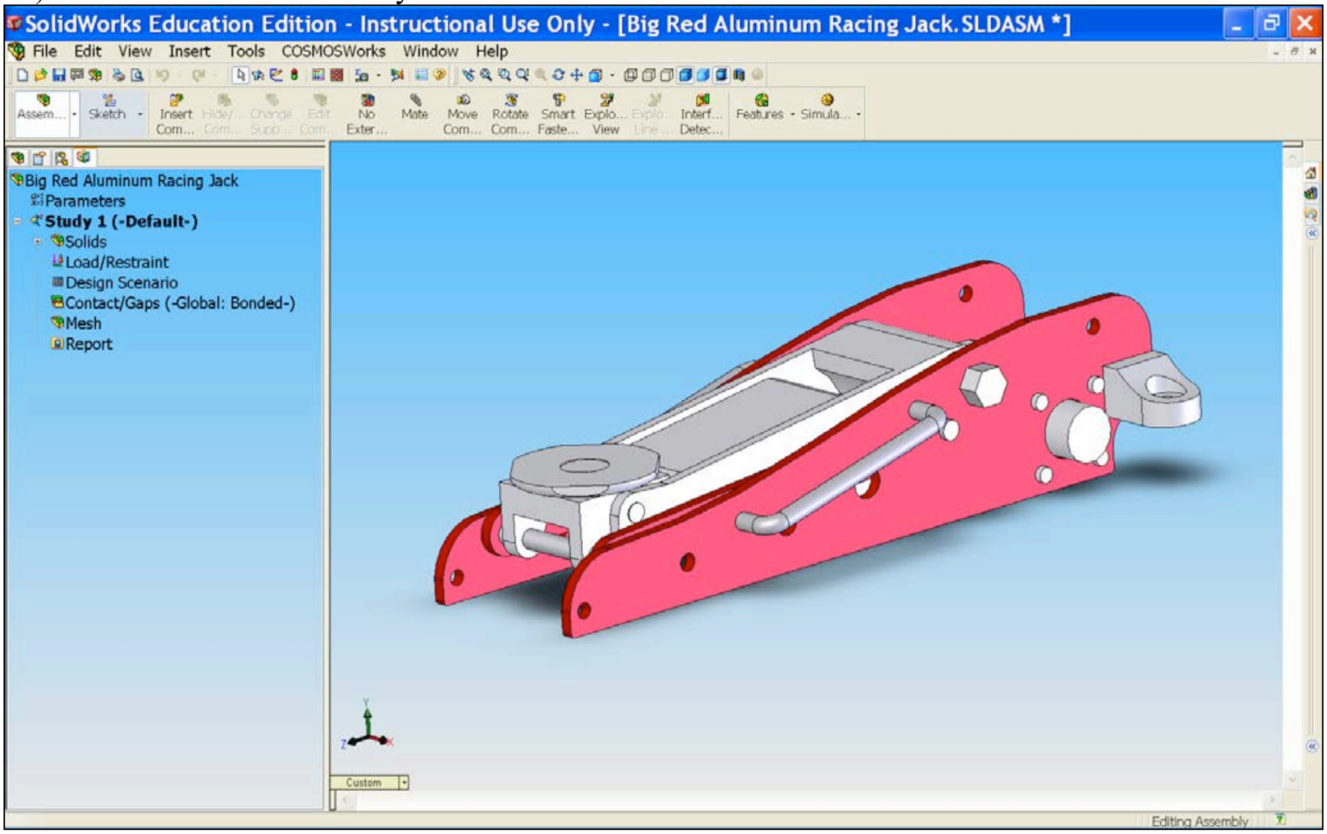
9) Right-click on Big Red Aluminum Jack and select **Study**.



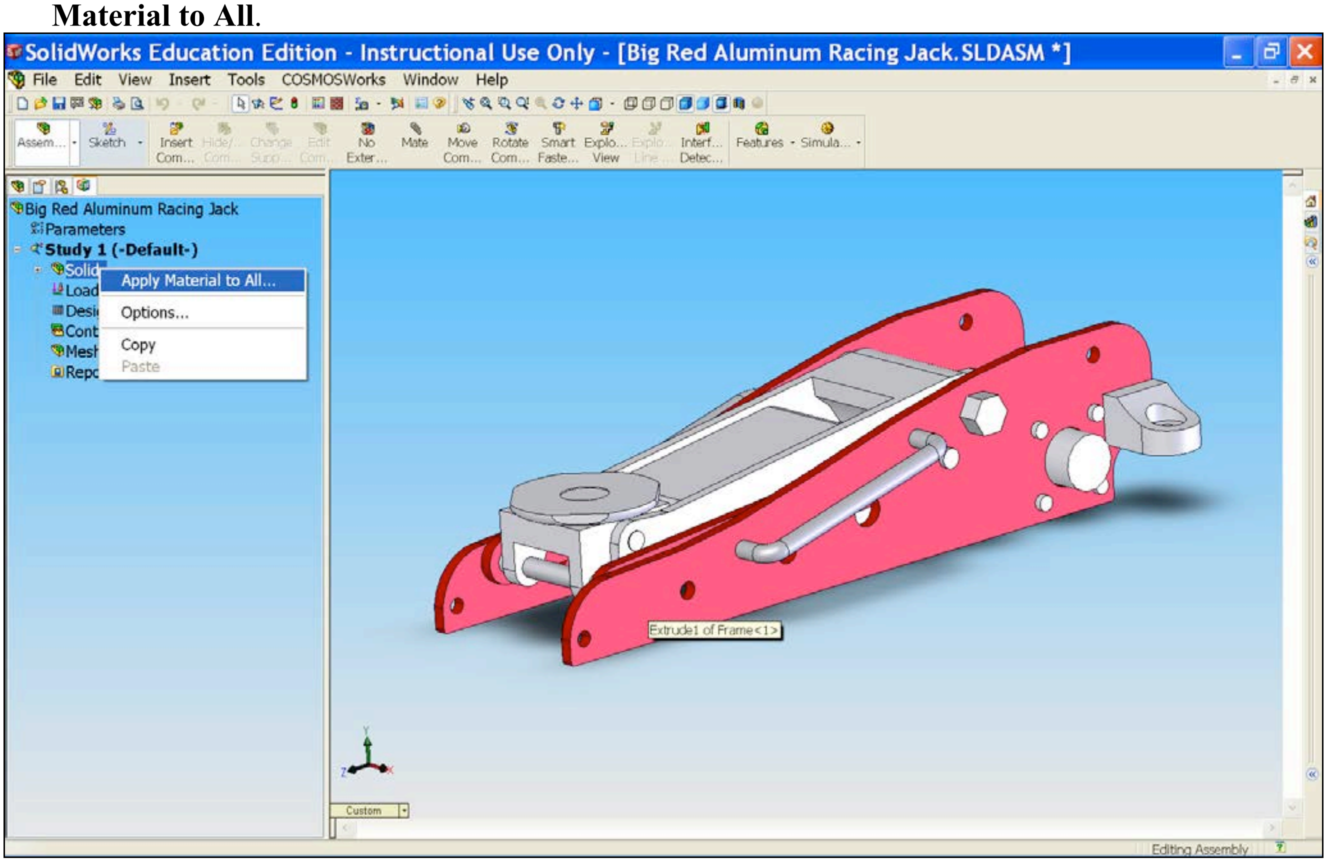
10) The default name of the study is Study 1. For Mesh Type, make sure that it is a solid mesh. Under Type, select  Static. Click OK to proceed.



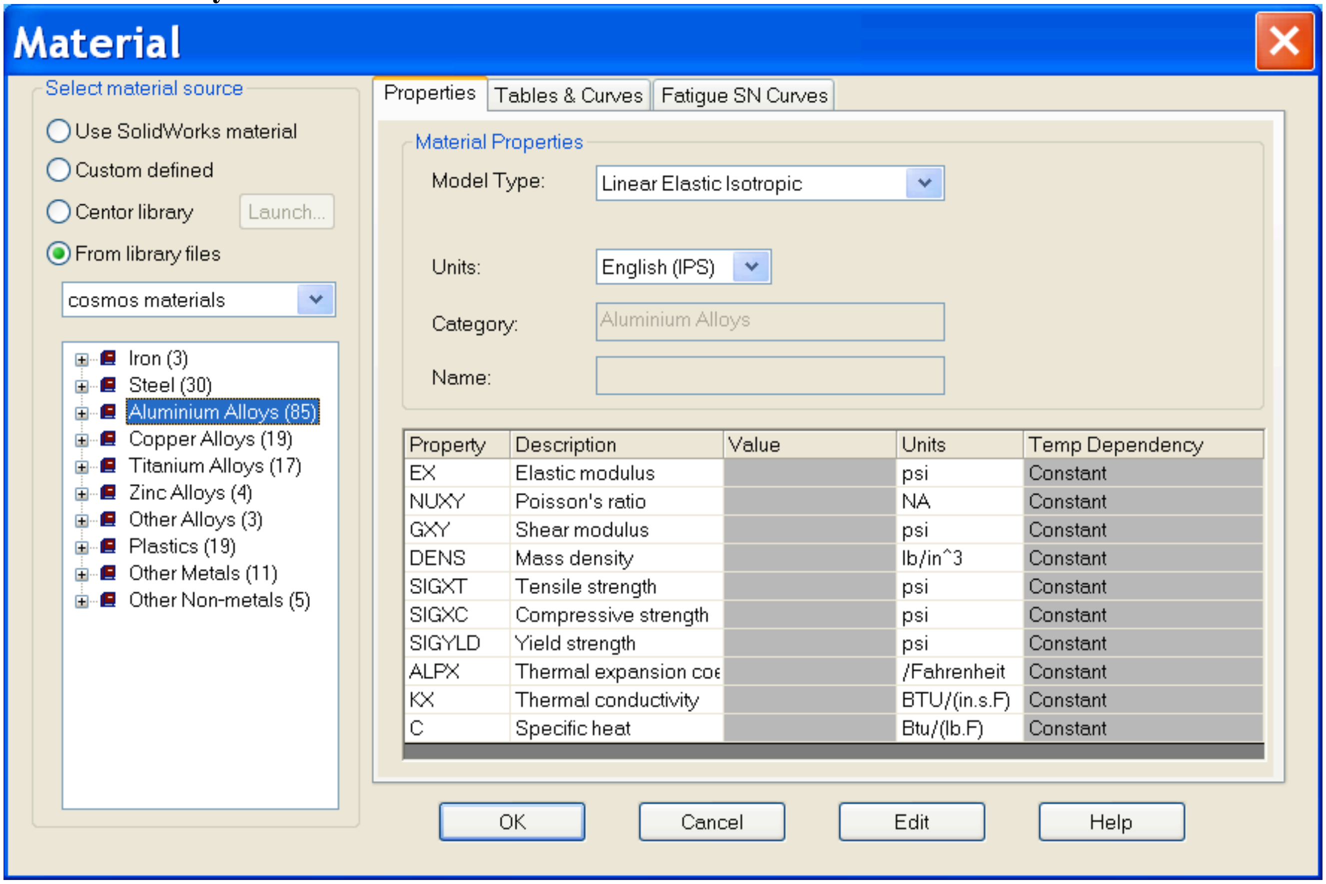
11) The COSMOSWorks study is now created.



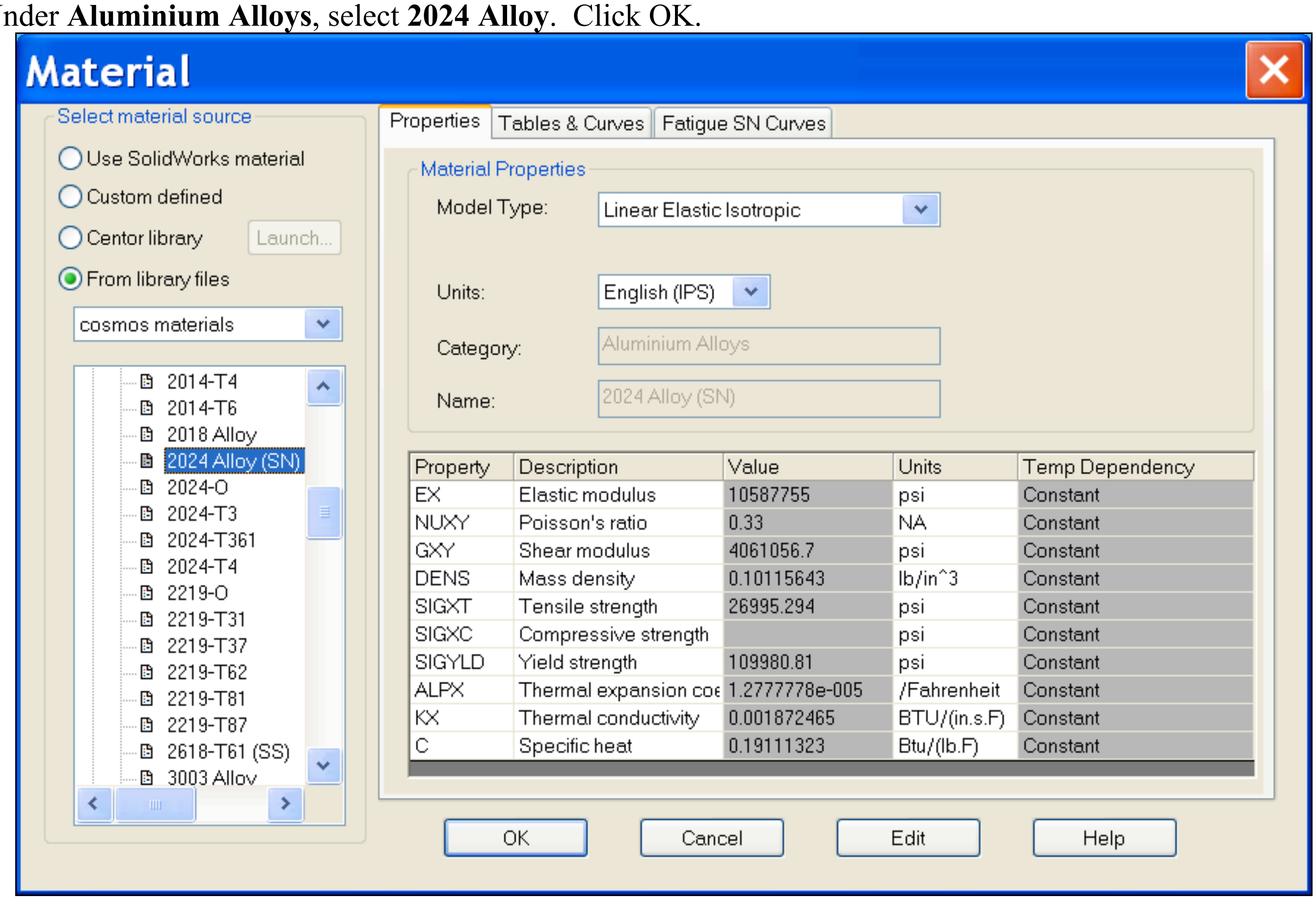
12) Now, we need to assign material for the floor jack assembly. Under Study1, right-click Solids and select **Apply Material to All**.



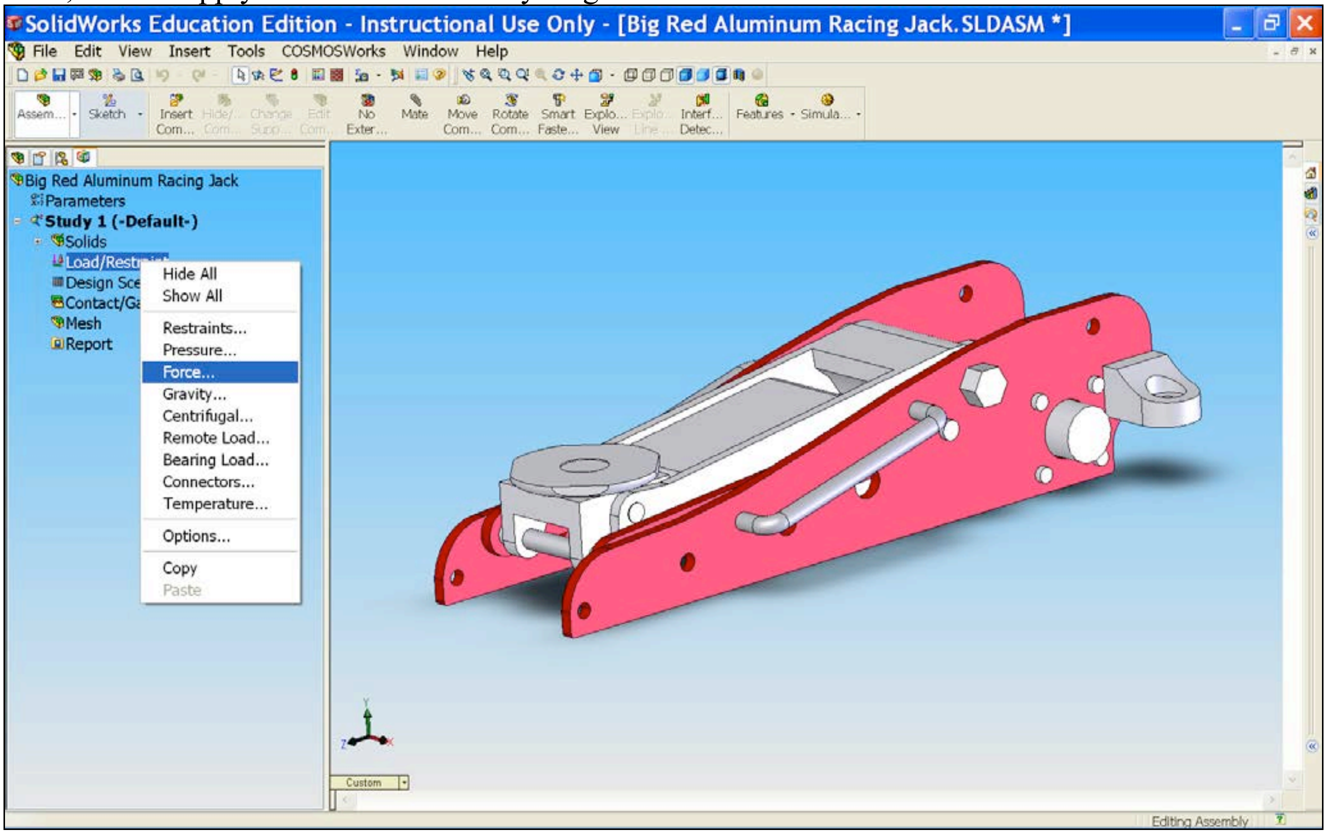
13) The material window pops up. Under **Select material source**, select **From library files**. Click the + sign next to **Aluminum Alloys**.



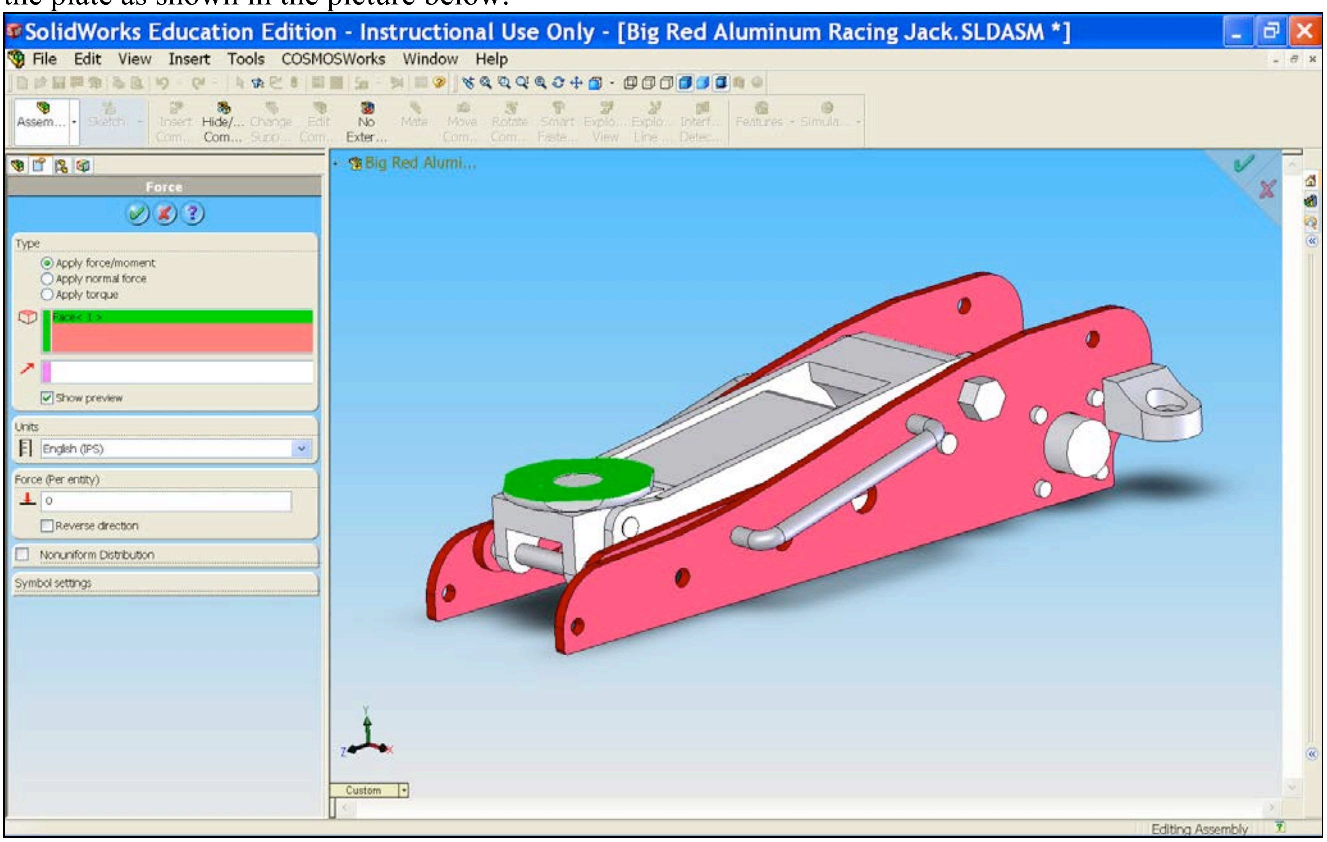
14) Under **Aluminium Alloys**, select **2024 Alloy**. Click OK.



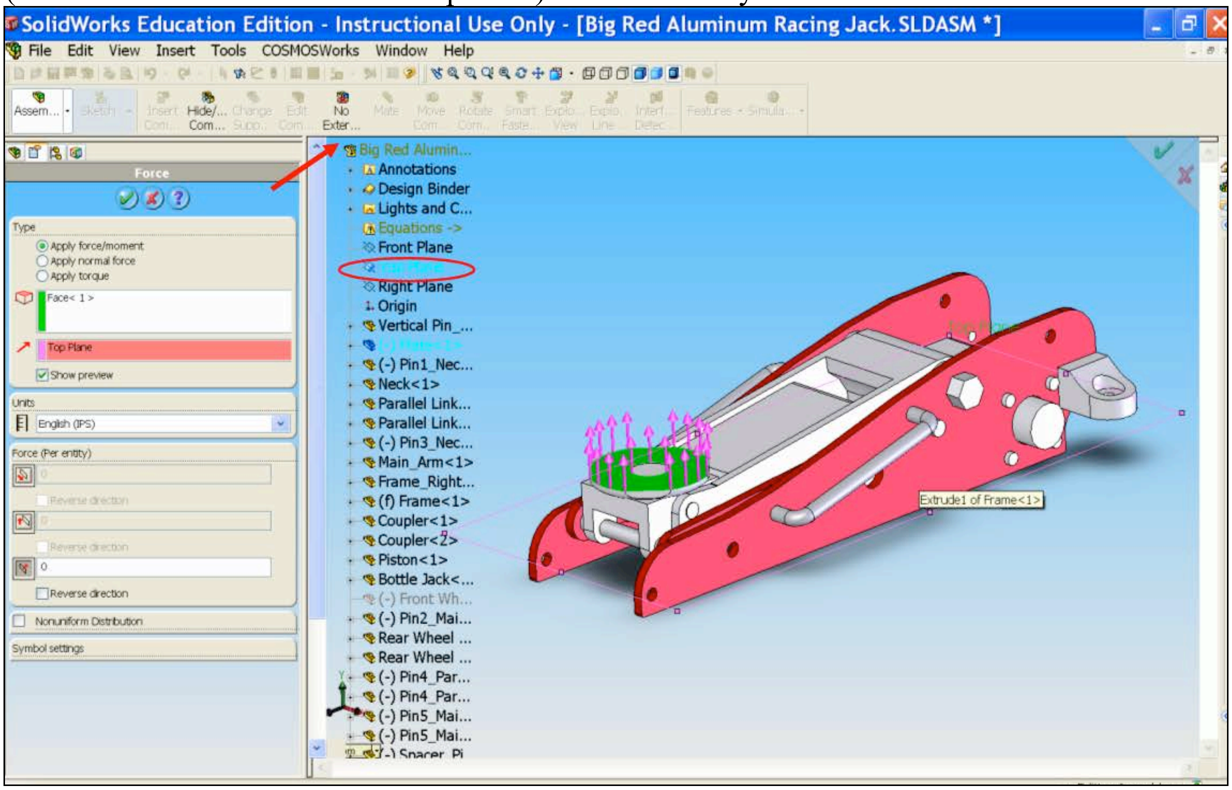
15) Now, we will apply force to the assembly. Right-click **Load/Restraint** select **Force**.



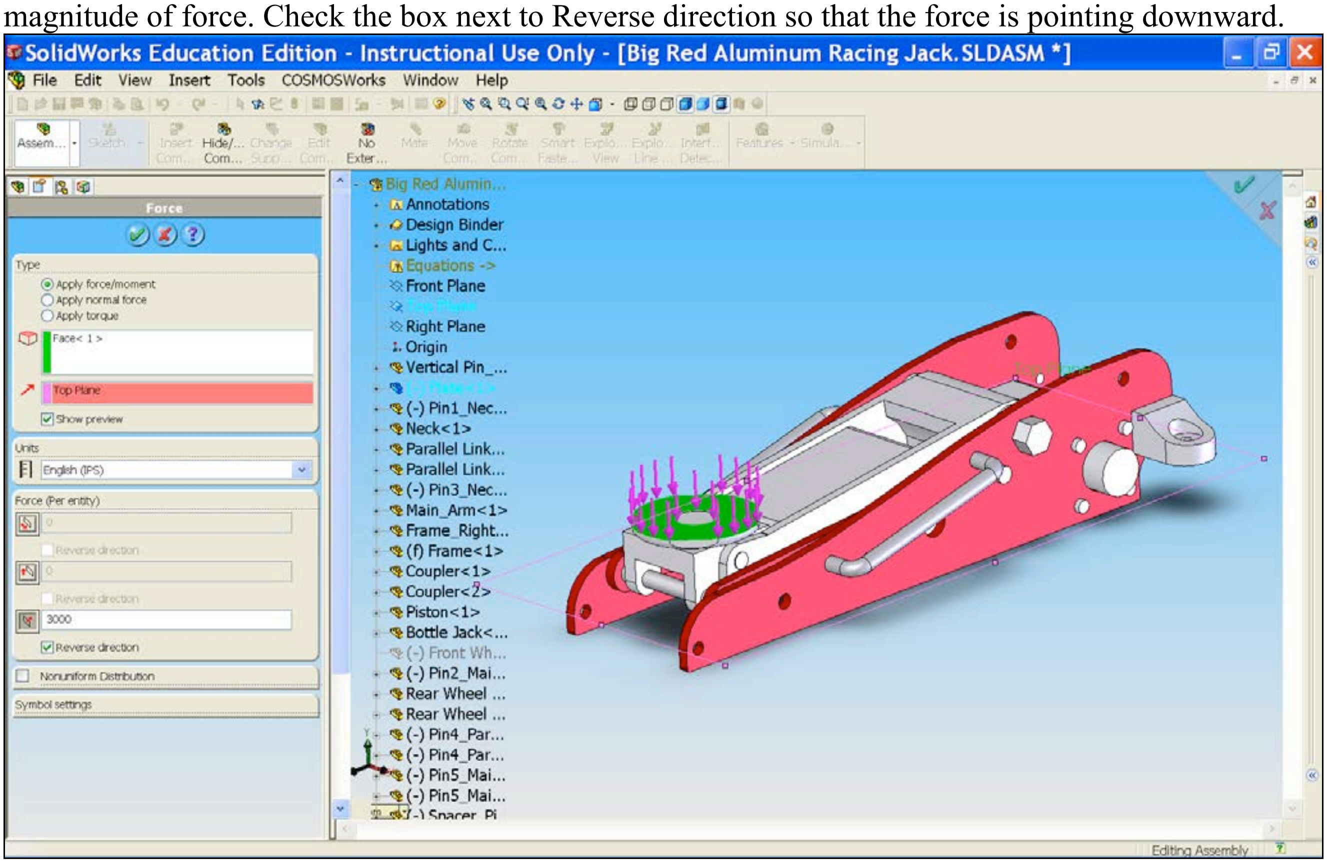
16) Under type, select **Apply force/moment**. For  **Faces, Edges, Vertices for Force**, select the top surface of the plate as shown in the picture below.



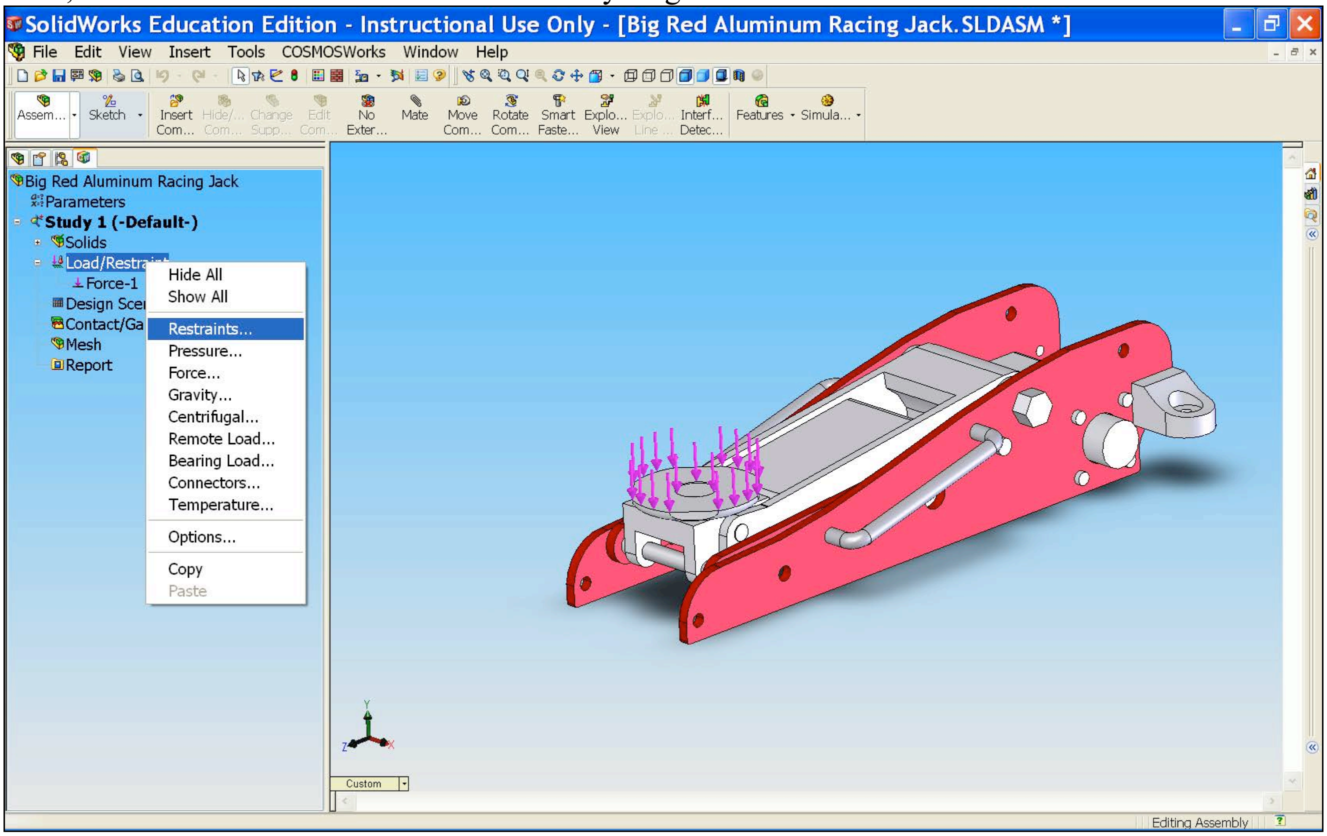
17) For **Face, Edge, Plane, Axis for Direction**, select the top plane of the assembly. To do that, click the + sign (pointed by the arrow in the picture) next to the assembly tree. When the assembly tree is displayed, select **Top Plane** (shown inside the red circle in the picture) of the assembly.



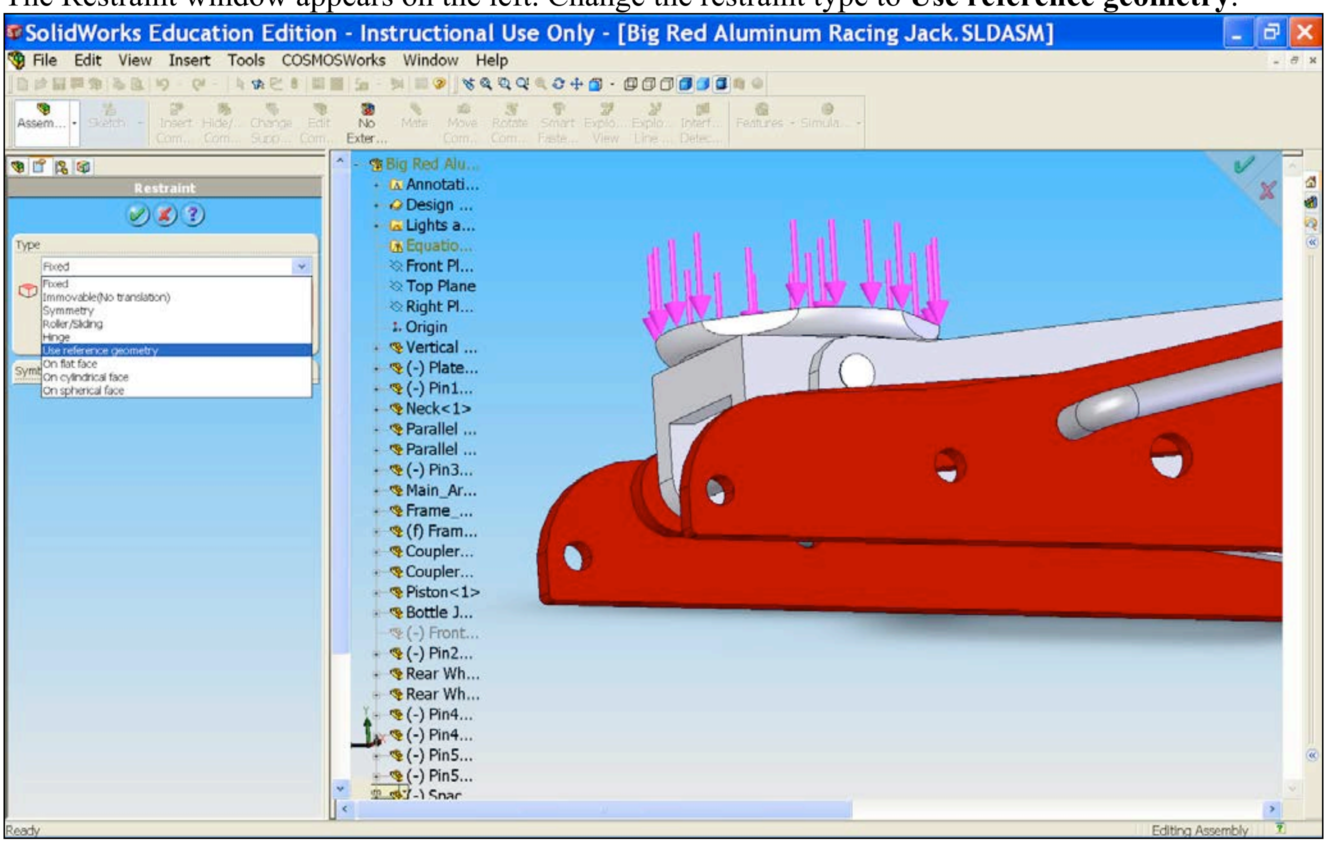
18) Change the **Units** to English (IPS). Under **Force**, select **normal to direction** and set **3000** as the magnitude of force. Check the box next to Reverse direction so that the force is pointing downward.

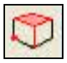


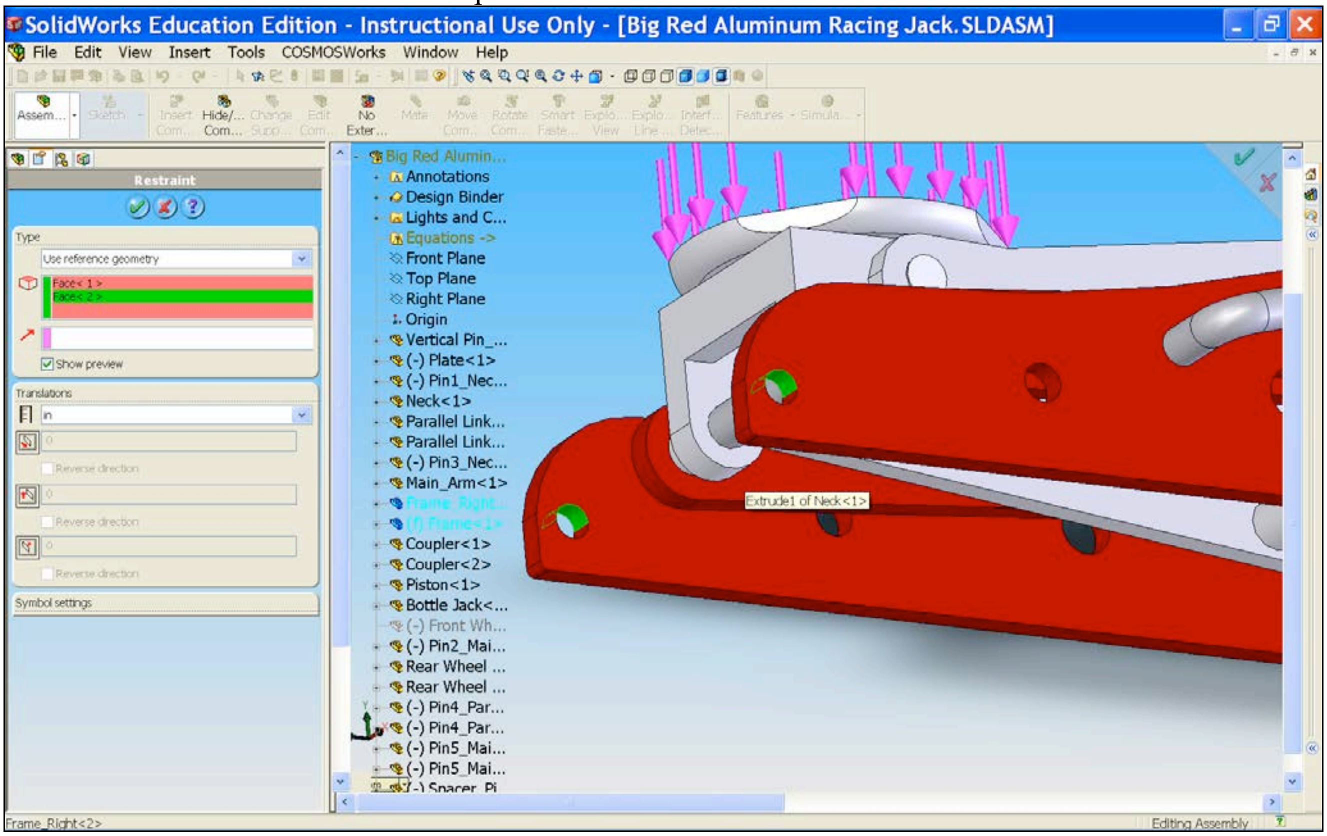
19) Now, we will define constraints to the assembly. Right-click  **Load/Restraint** and select **Restraints**.



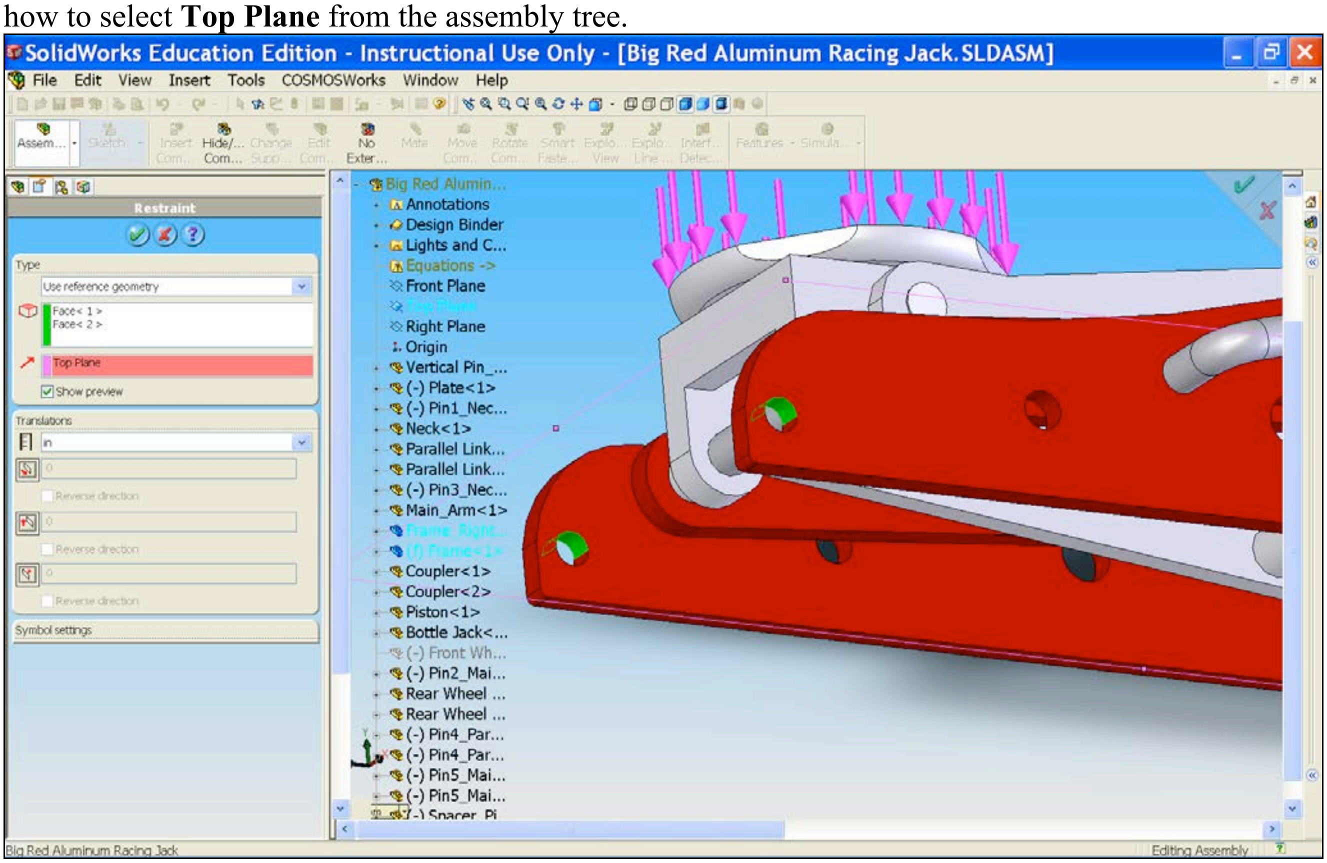
20) The Restraint window appears on the left. Change the restraint type to **Use reference geometry**.



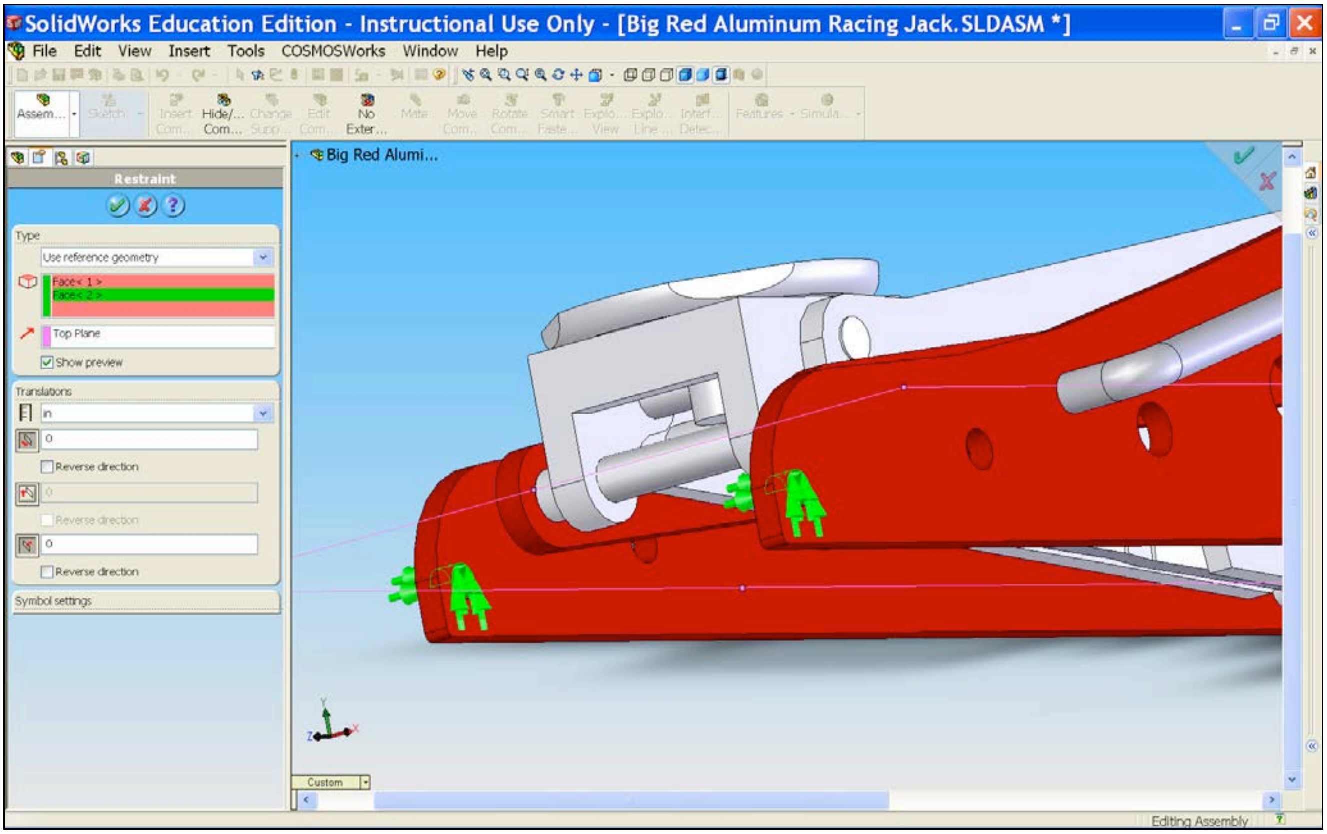
21) For  **Faces, Edges, Vertices for Restraint**, select two upper half of the cylinder where the shaft for the front wheel is mounted. Refer to the picture below.



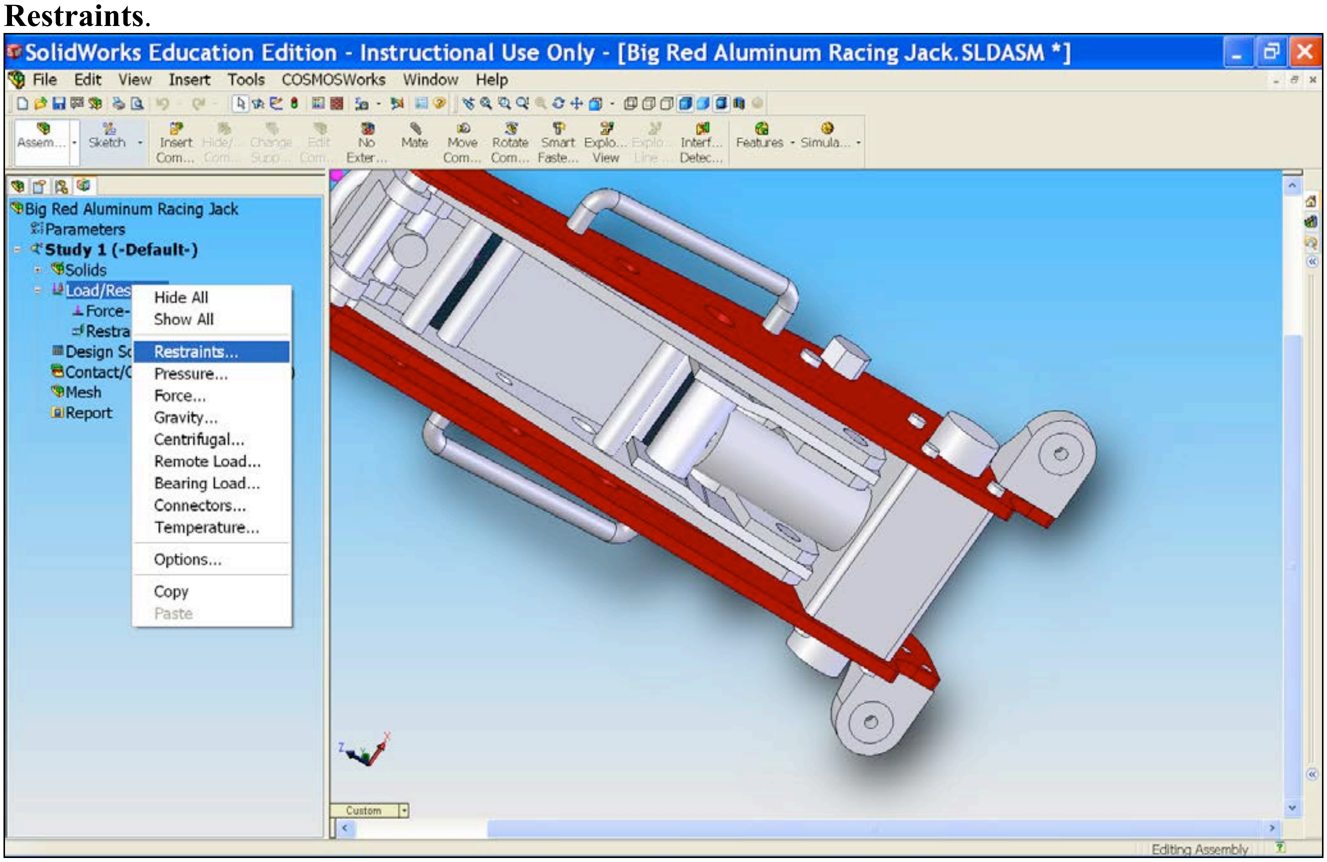
22) Select Top Plane of the assembly as the  **direction for restraint**. Refer to step 17 if you are still not sure of how to select **Top Plane** from the assembly tree.



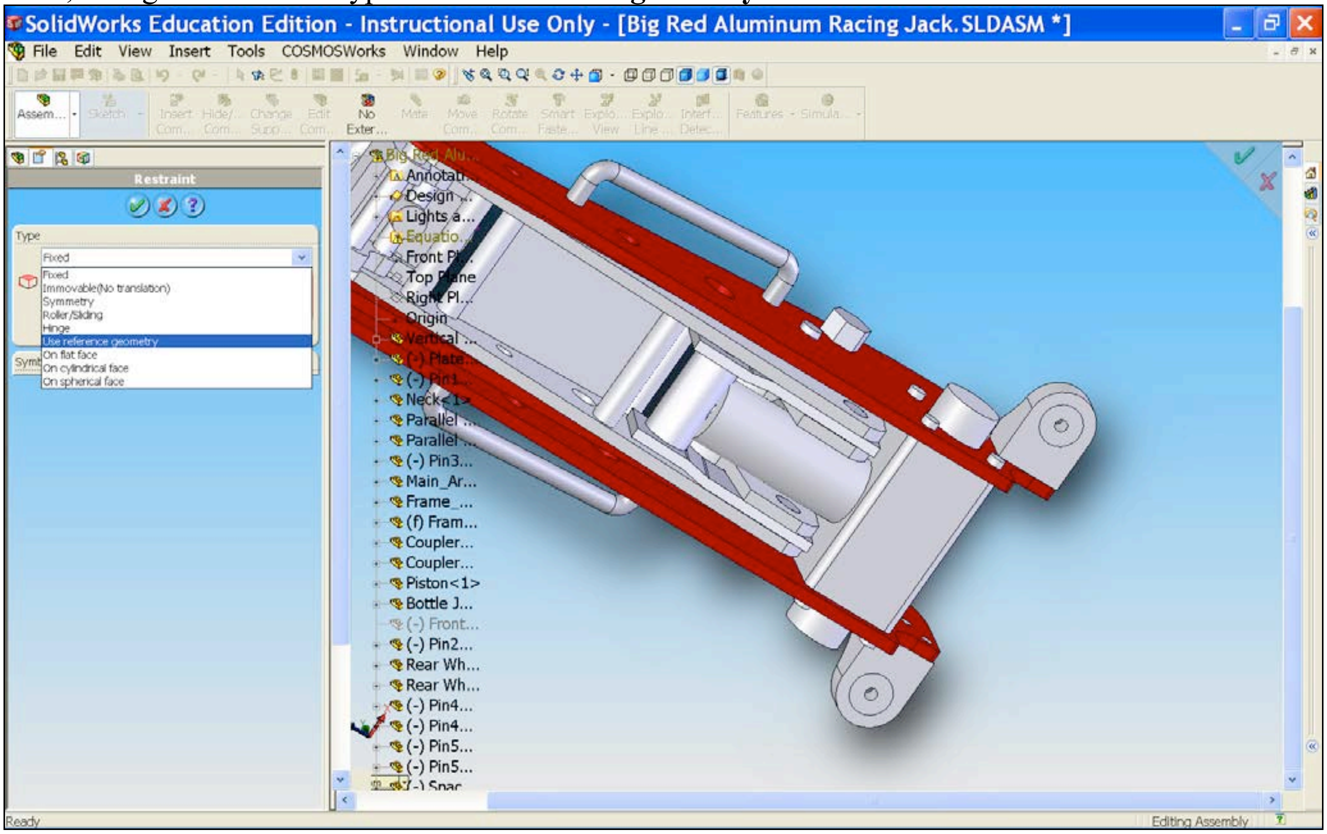
23) Under translation, select  **Along plane Dir 1** and  **Normal to plane** and set the value to 0 for both of them.

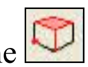


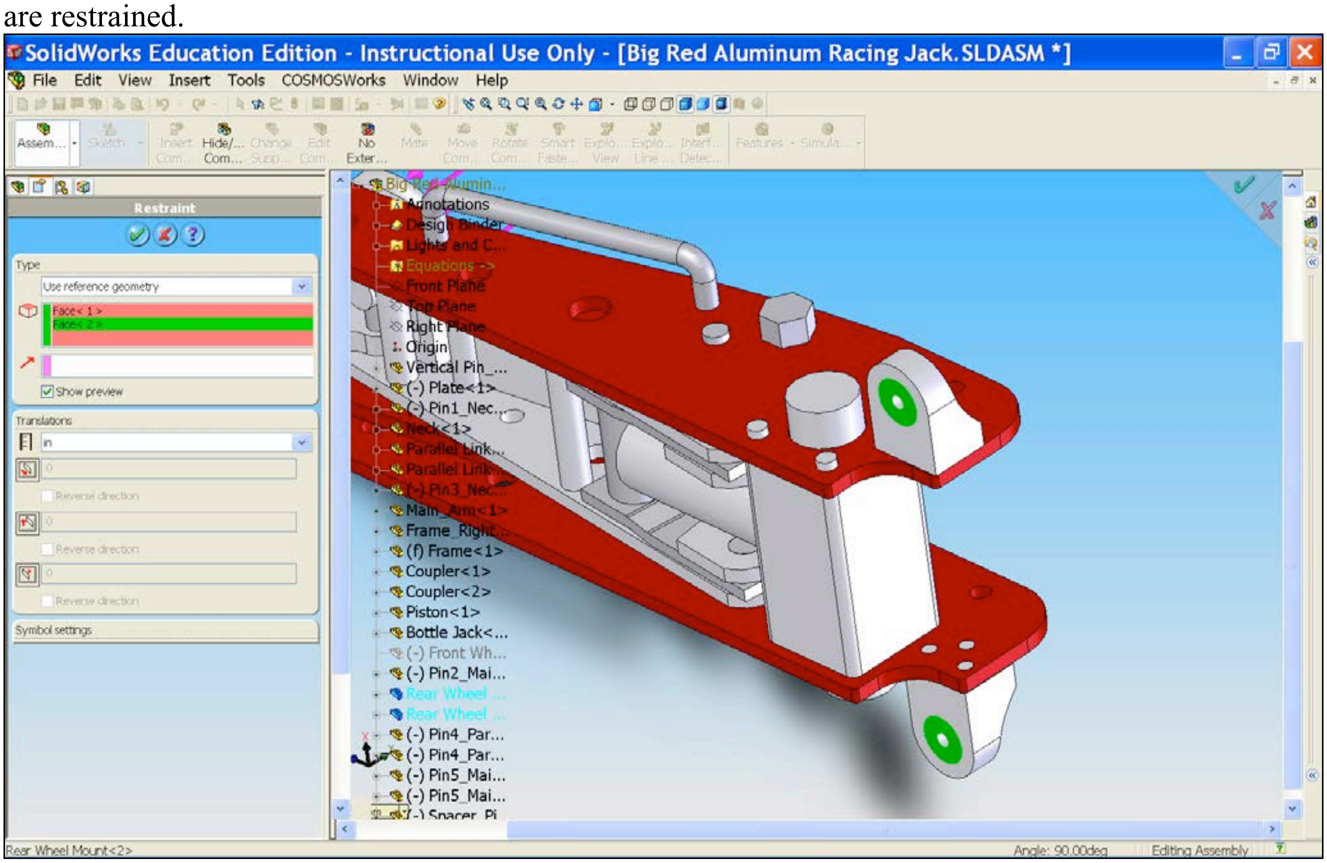
24) Now, we will define the second restraint to the assembly. Right-click  **Load/Restraint** and select **Restraints**.



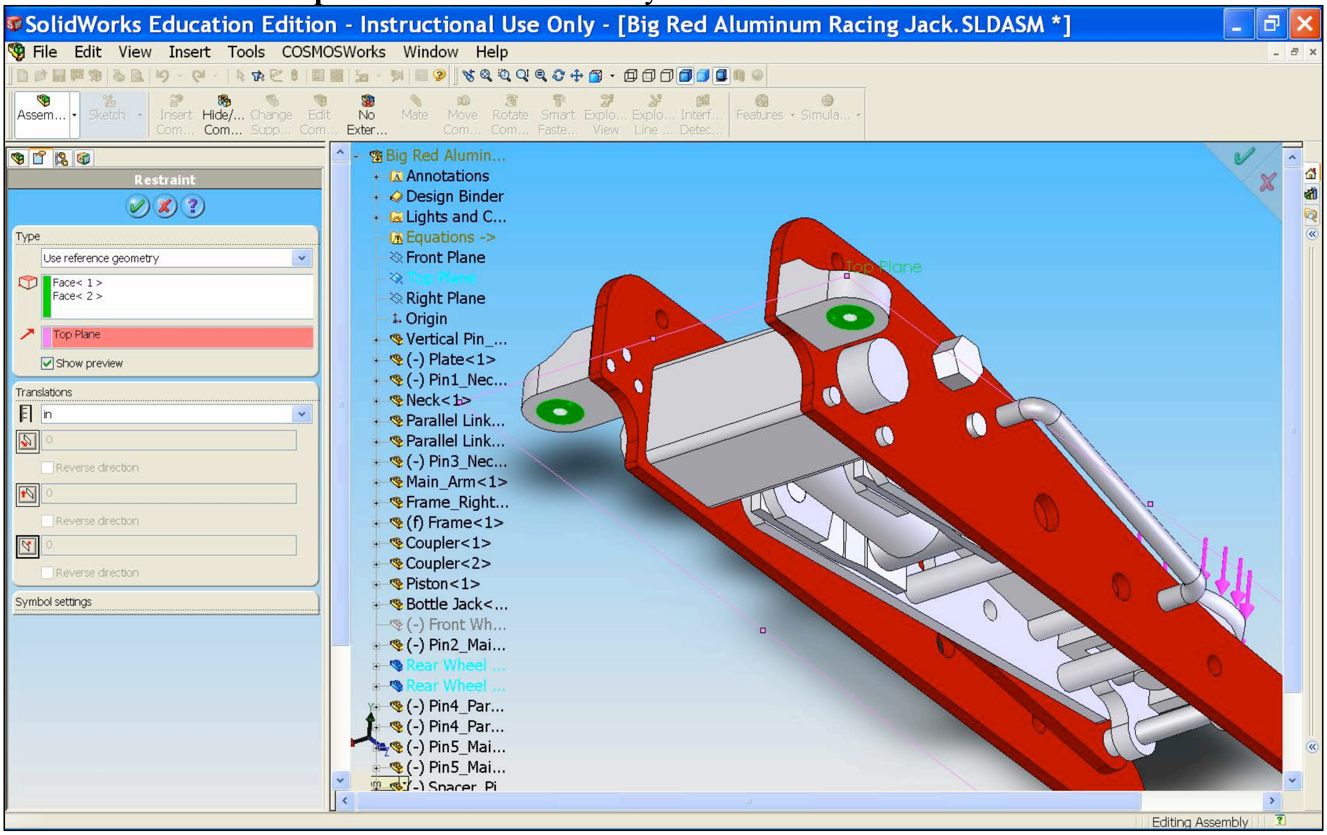
25) Next, change the restraint type to **Use reference geometry**.

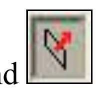


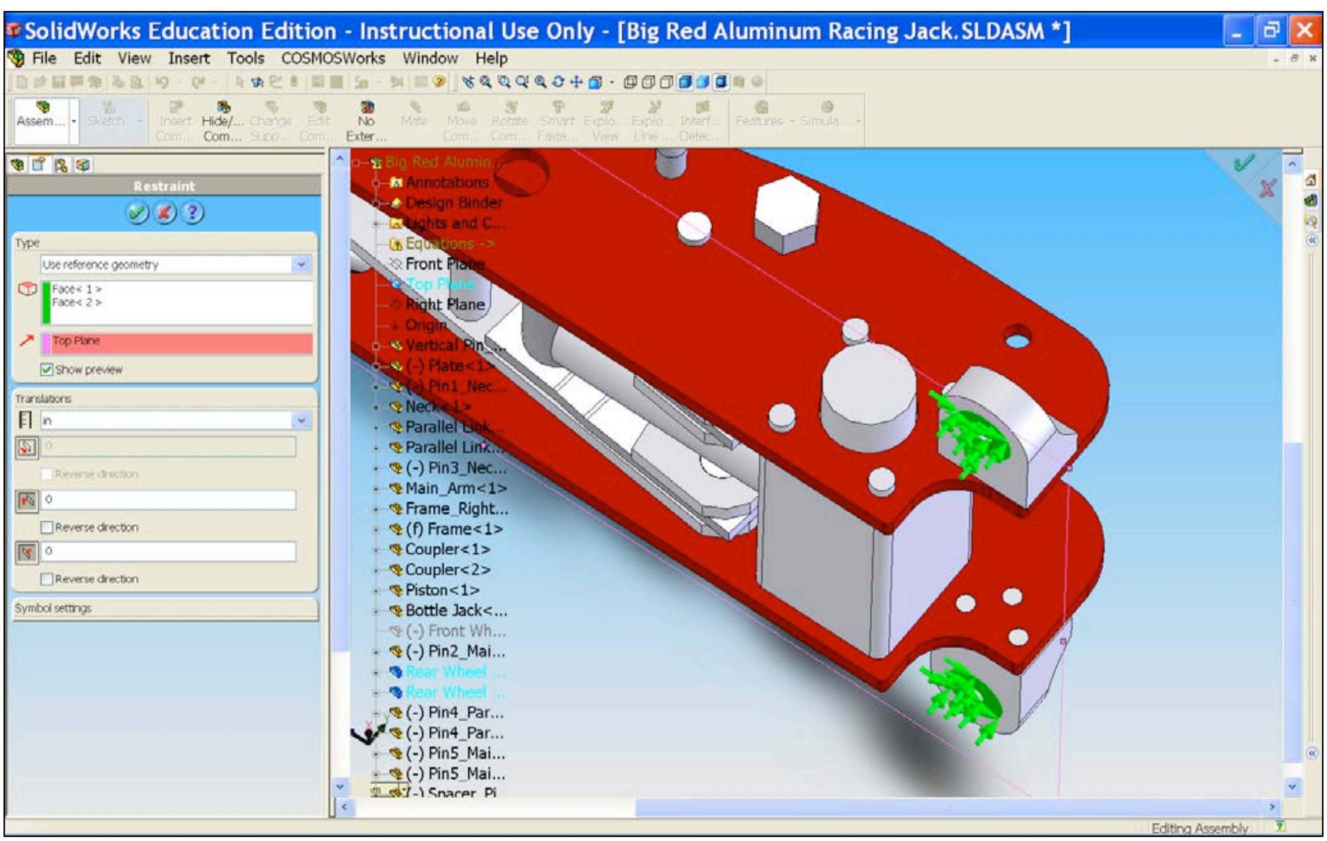
26) Under the rear wheel mounts, click on the two circular faces as shown in the picture for the  **Faces, Edges, Vertices for Restraint**. These two faces are in contact with the bearings from the rear wheels. Therefore, they are restrained.



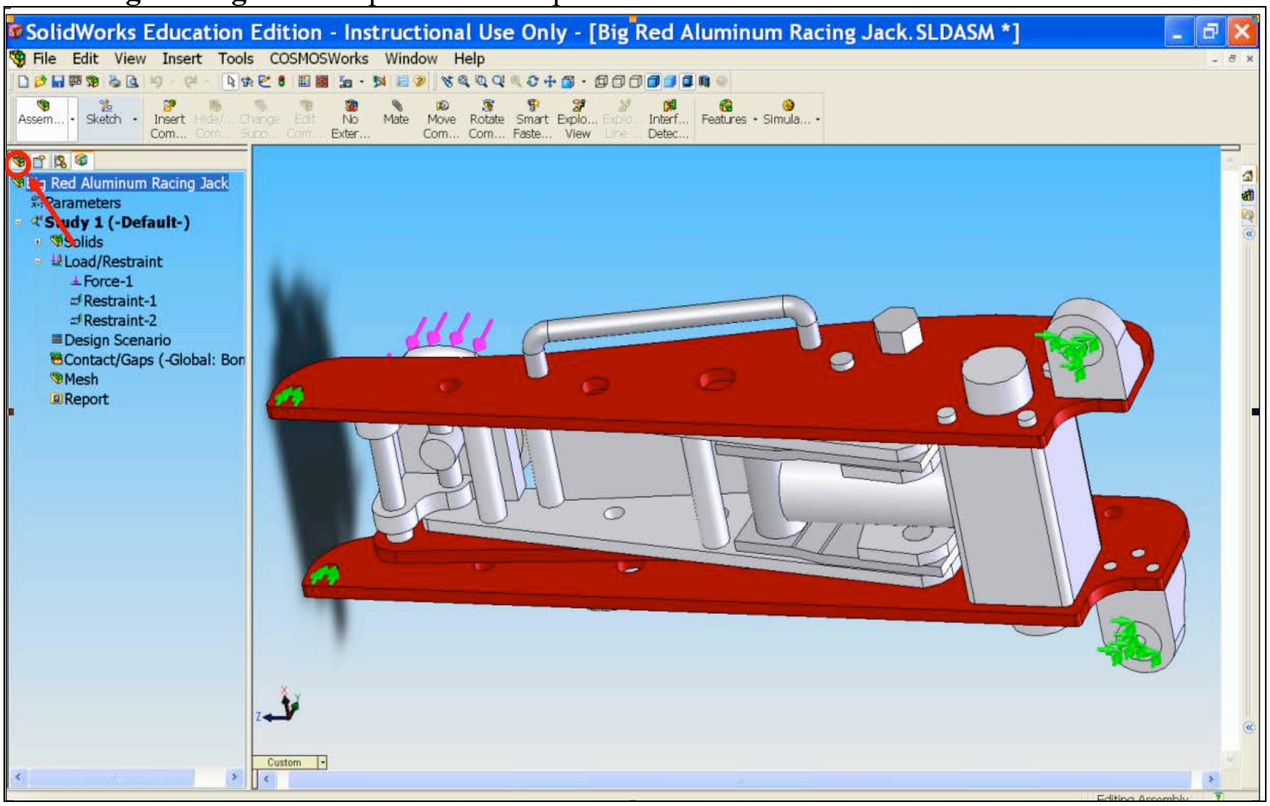
27) For the  **direction for restraint**, select the **Top Plane** of the assembly. Refer to step 17 if you are still not sure of how to select **Top Plane** from the assembly tree.



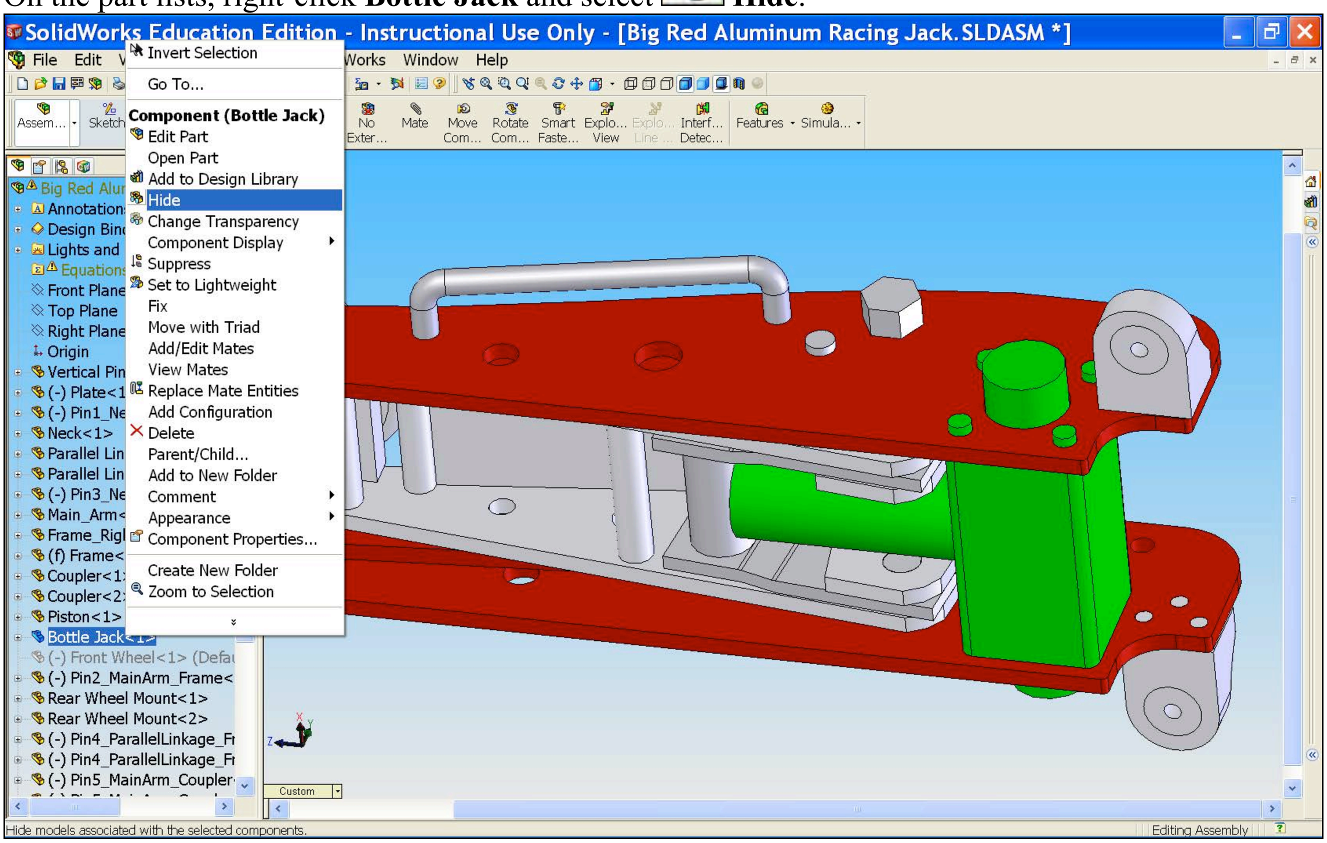
28) Under translation, select  **Along plane Dir 2** and  **Normal to plane** and set the value to 0 for both of them.



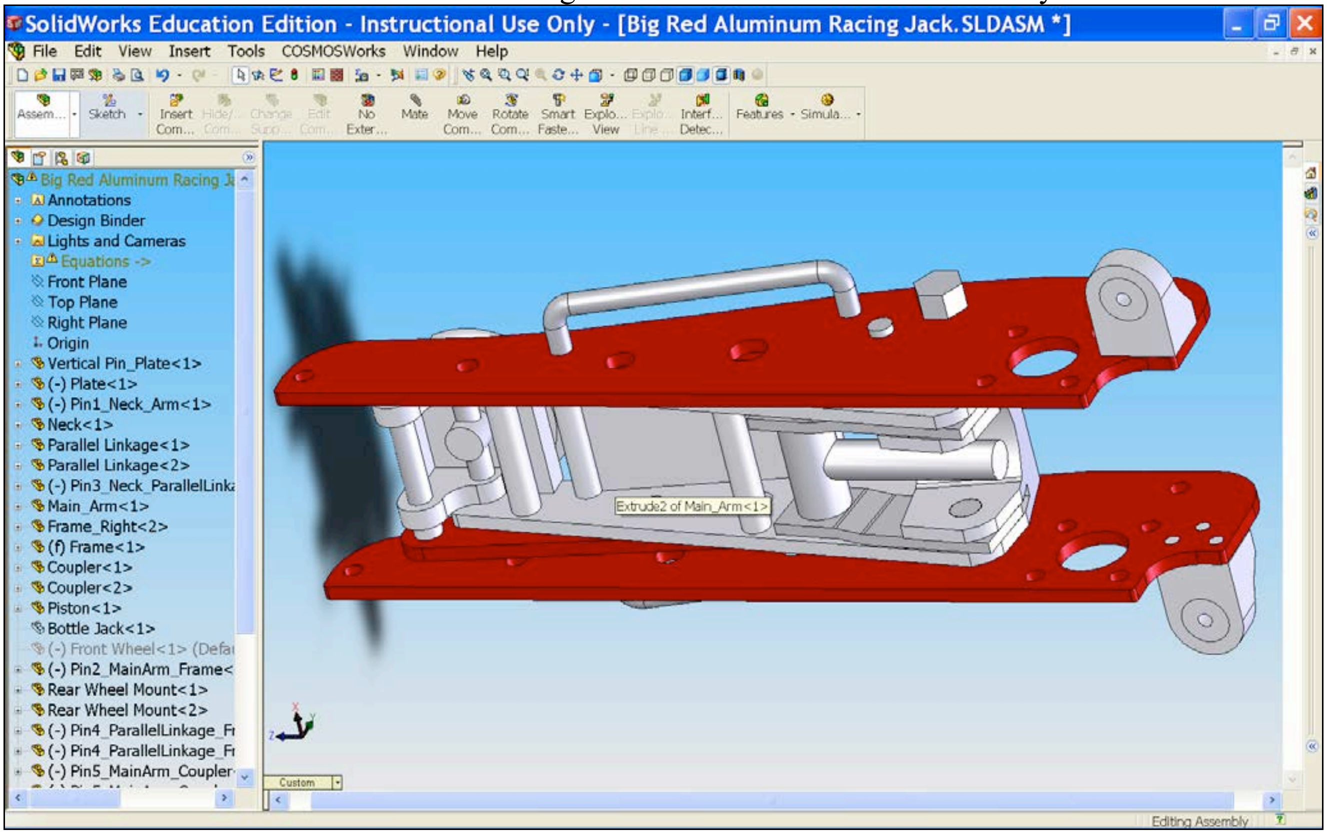
29) So far, we have constrained two parts of the assembly in which the wheels are mounted. Next, we need to constrain the end of piston. In order to access the end of piston, we must first hide the bottle jack. Go to **FeatureManager design tree** as pointed in the picture below.



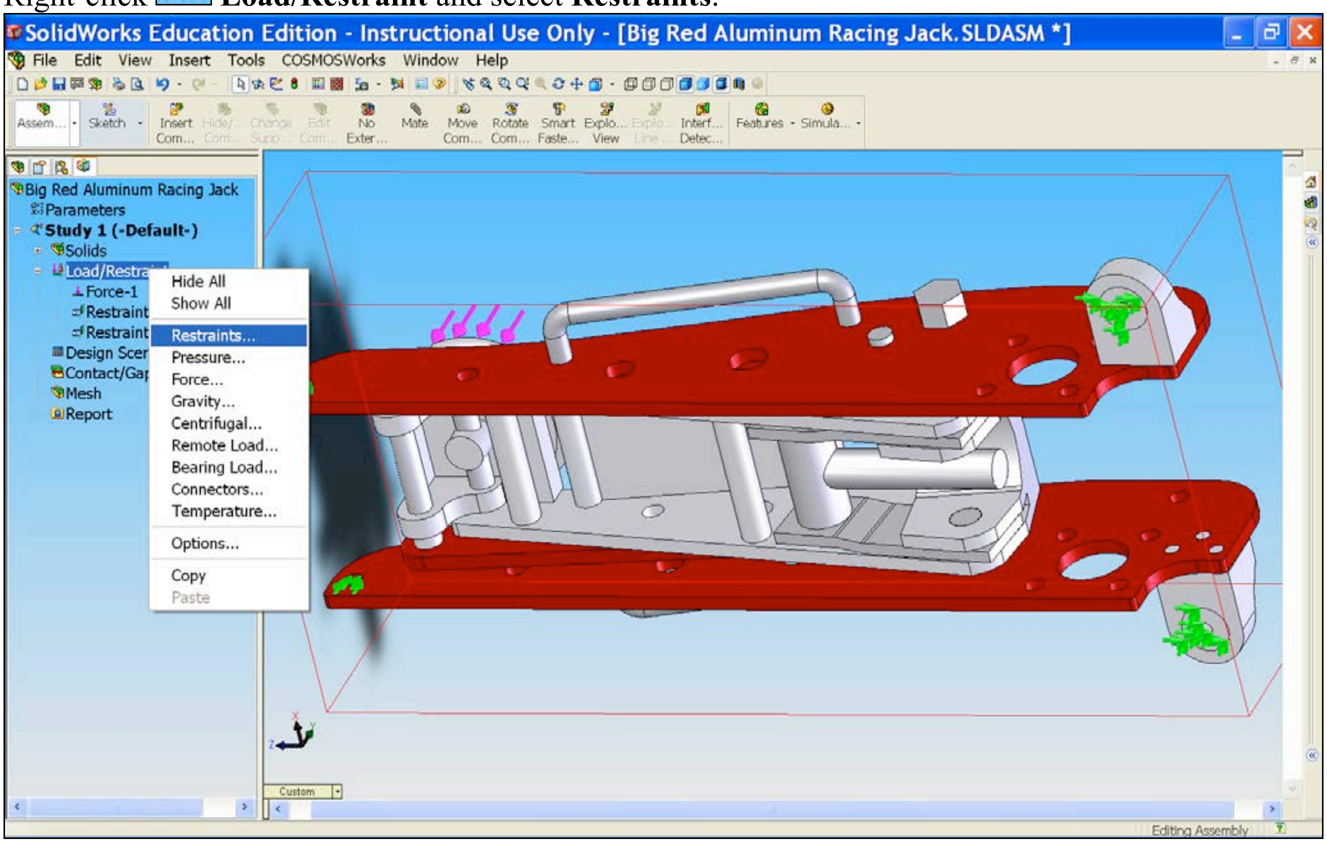
30) On the part lists, right-click **Bottle Jack** and select  **Hide**.



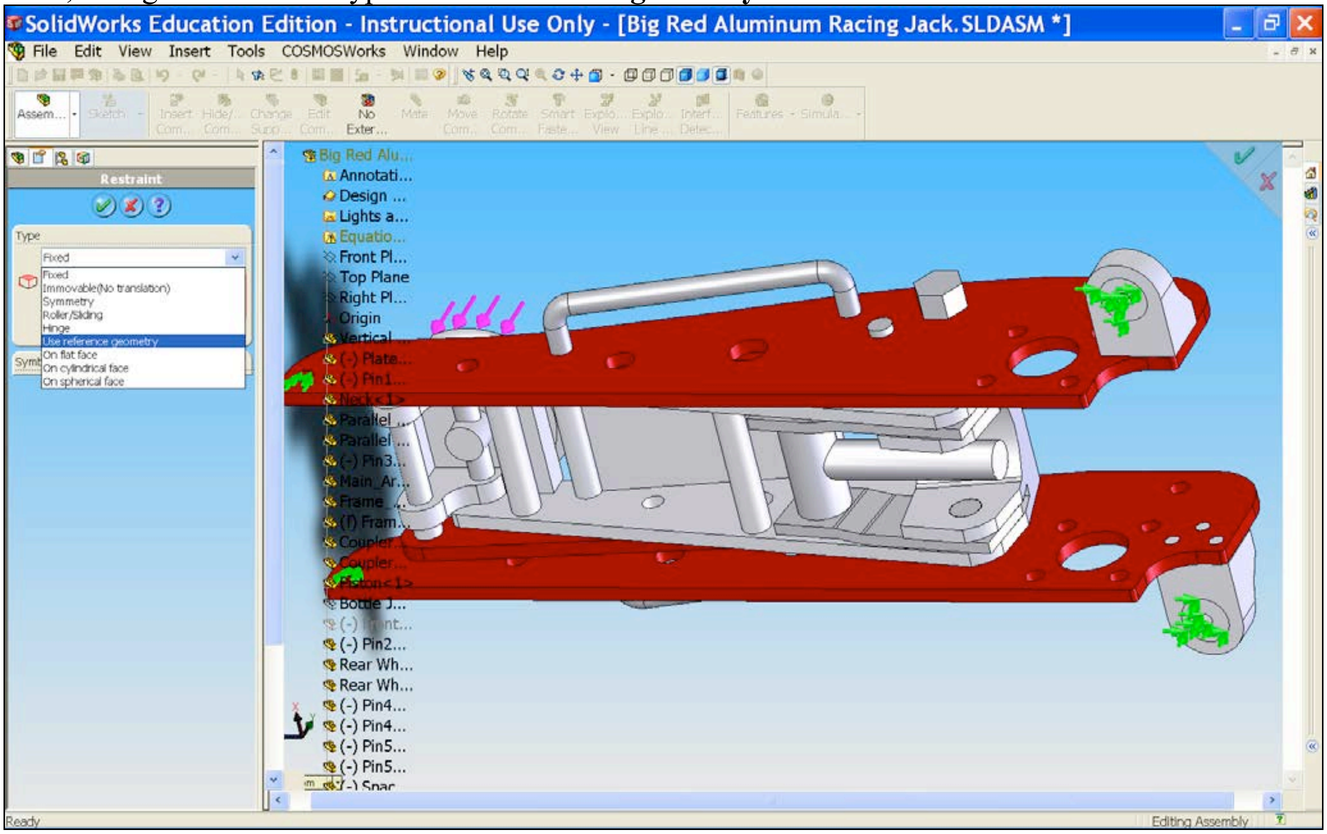
31) The Bottle Jack is now hidden and we can get back to the COSMOSWorks Study.



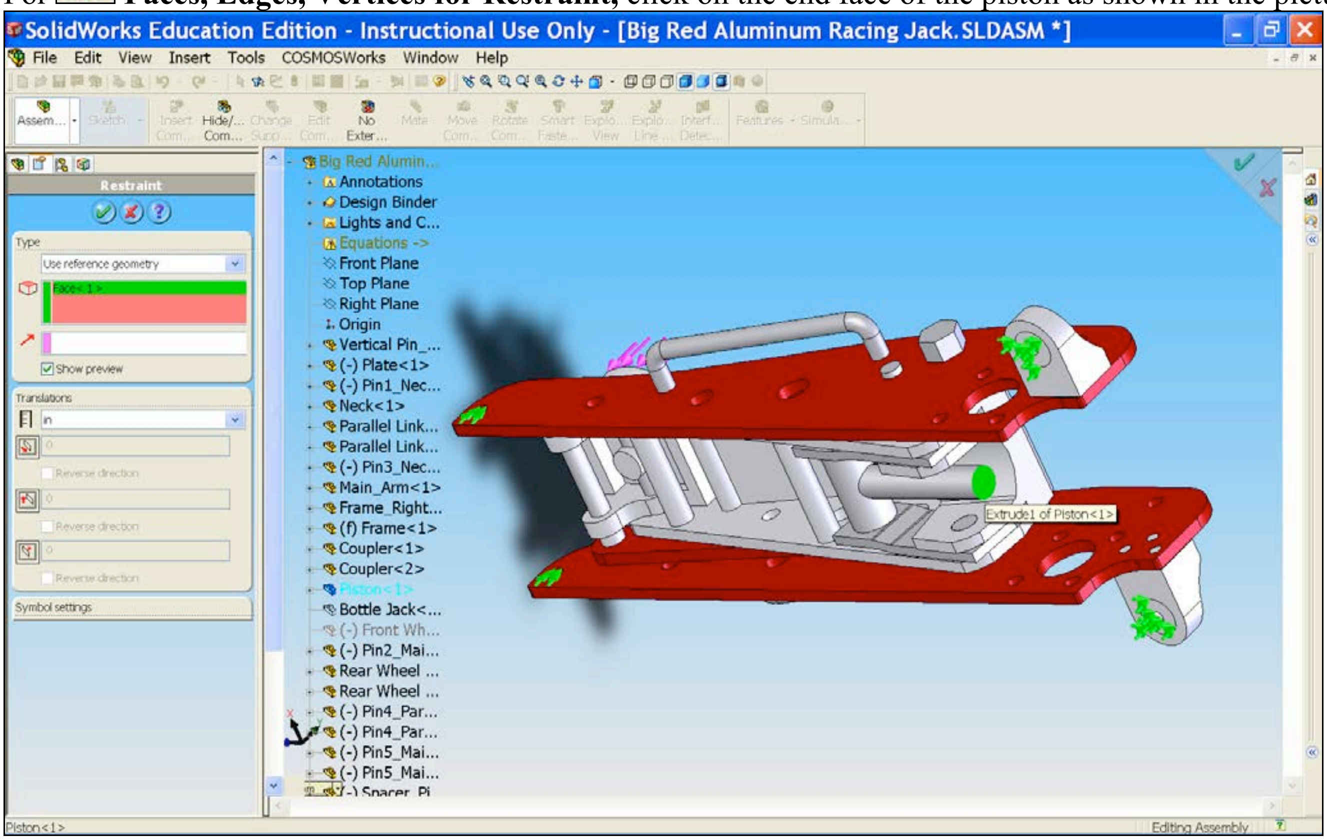
32) Right-click  **Load/Restraint** and select **Restraints**.



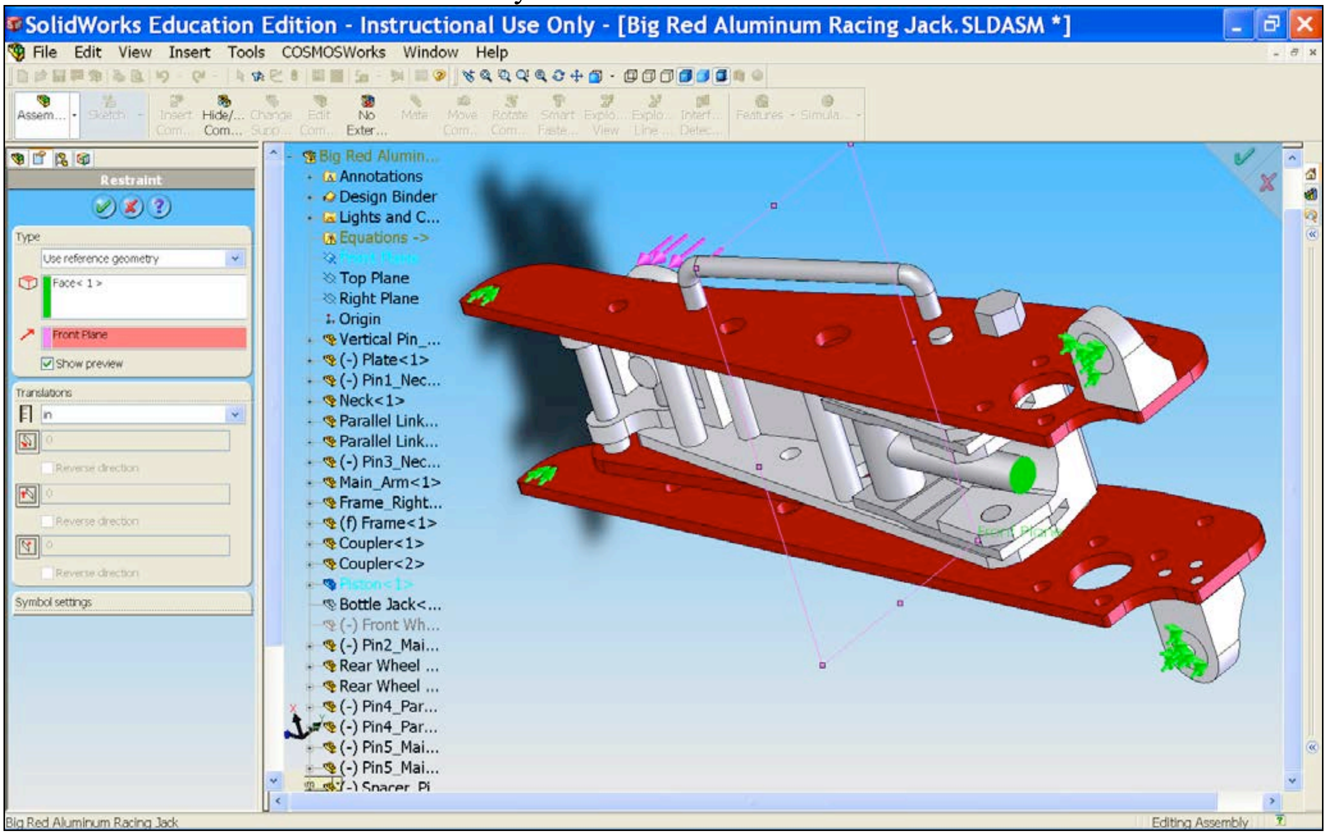
33) Next, change the restraint type to **Use reference geometry**.



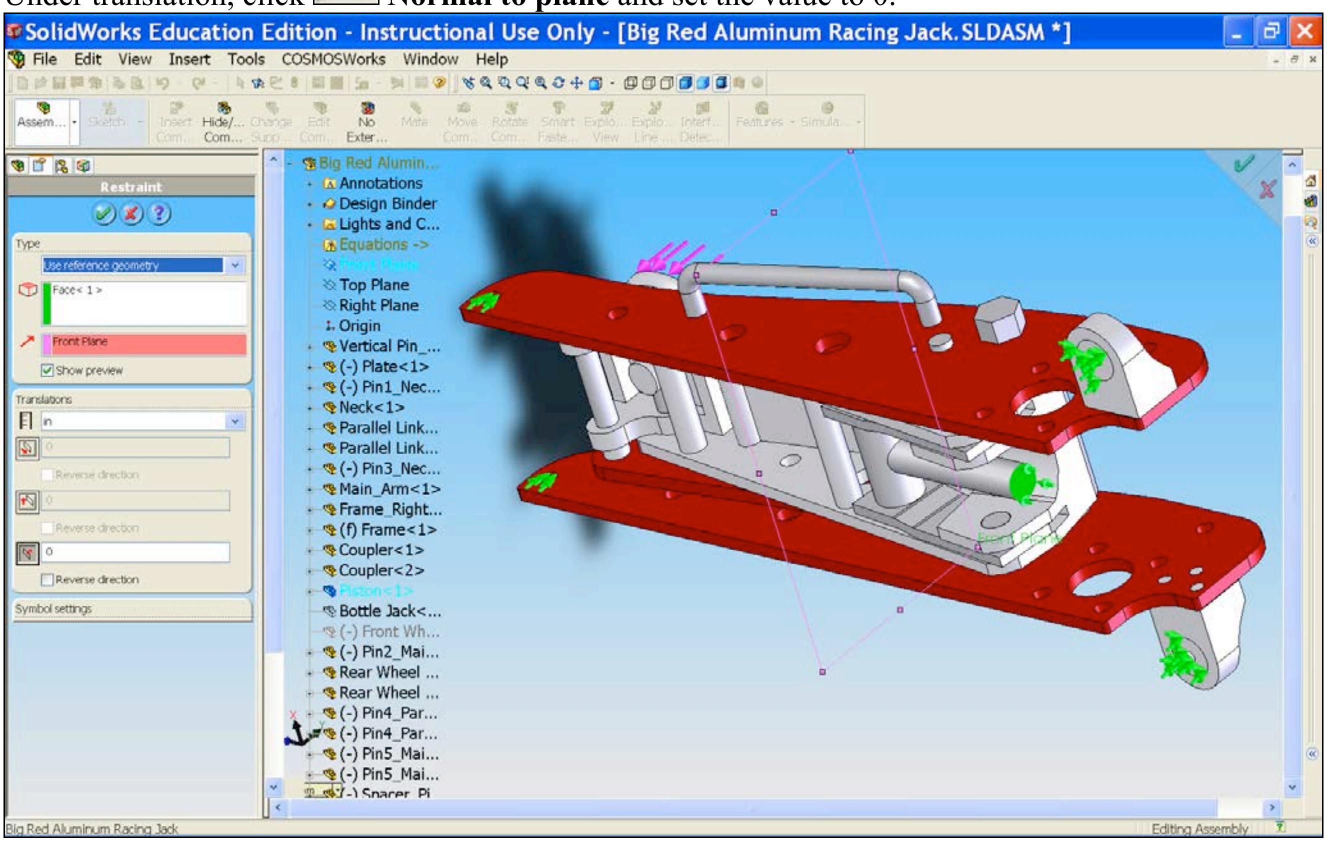
34) For  **Faces, Edges, Vertices for Restraint,** click on the end face of the piston as shown in the picture.



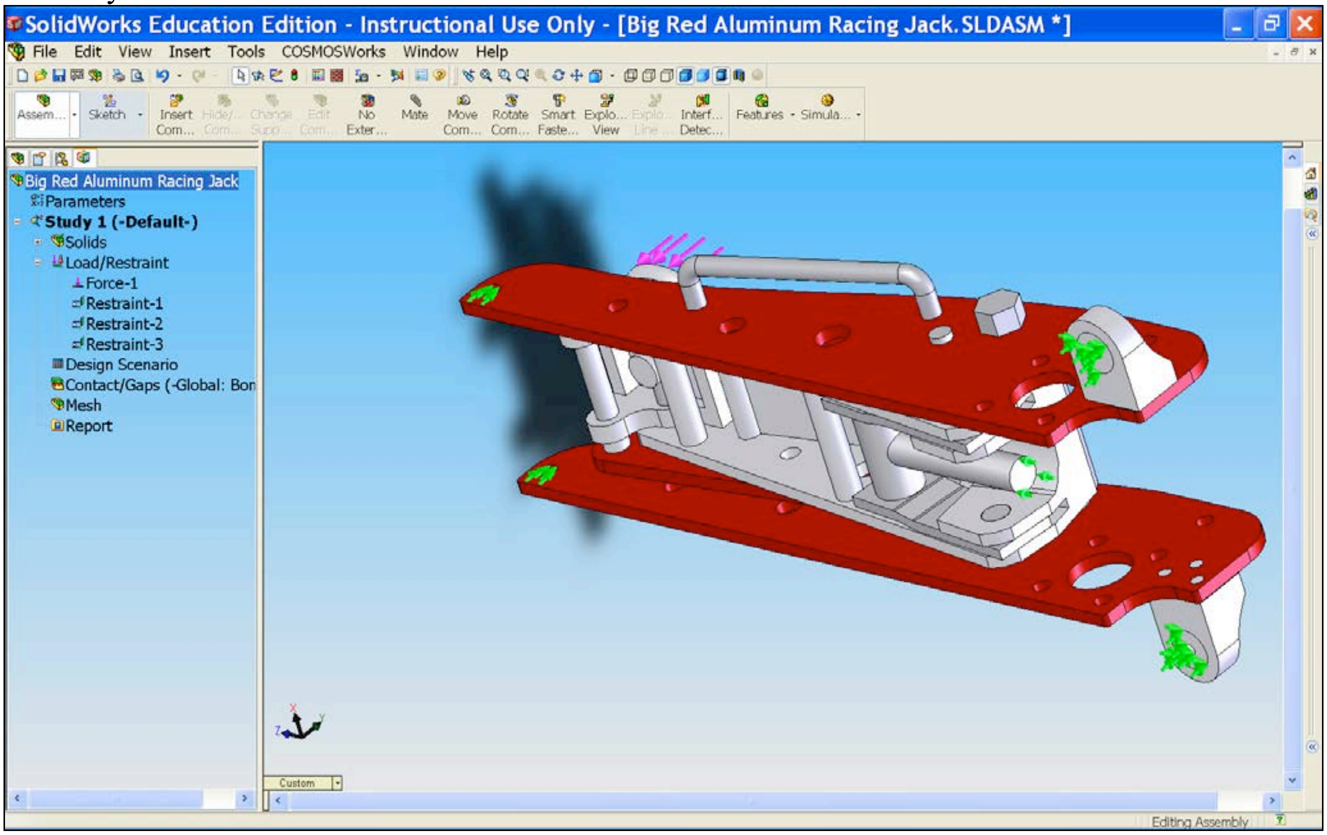
35) Select the Front Plane from the assembly tree as the  **direction for restraint**.



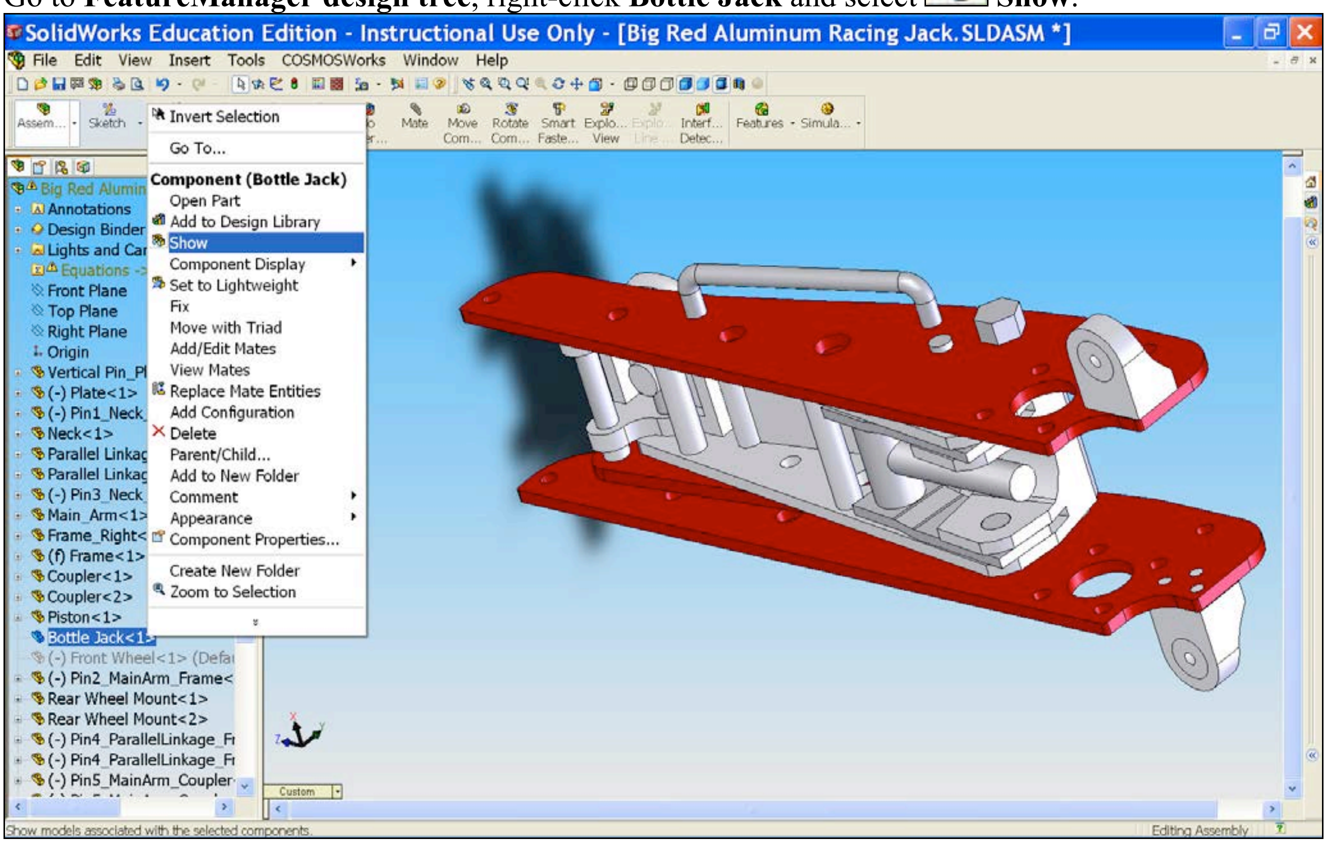
36) Under translation, click  **Normal to plane** and set the value to 0.



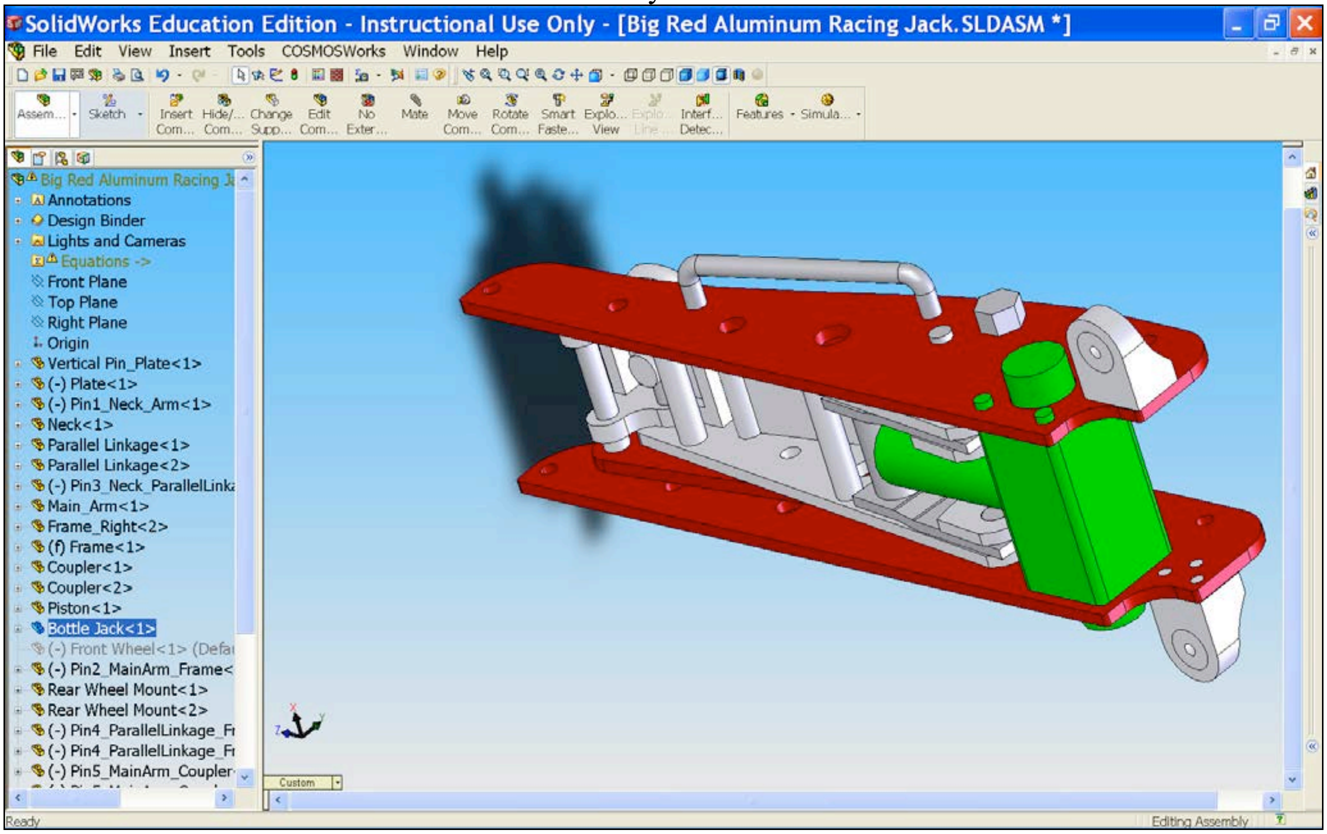
37) We have completed applying restraints to the assembly. Now, we can show the Bottle Jack back to the assembly.

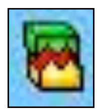


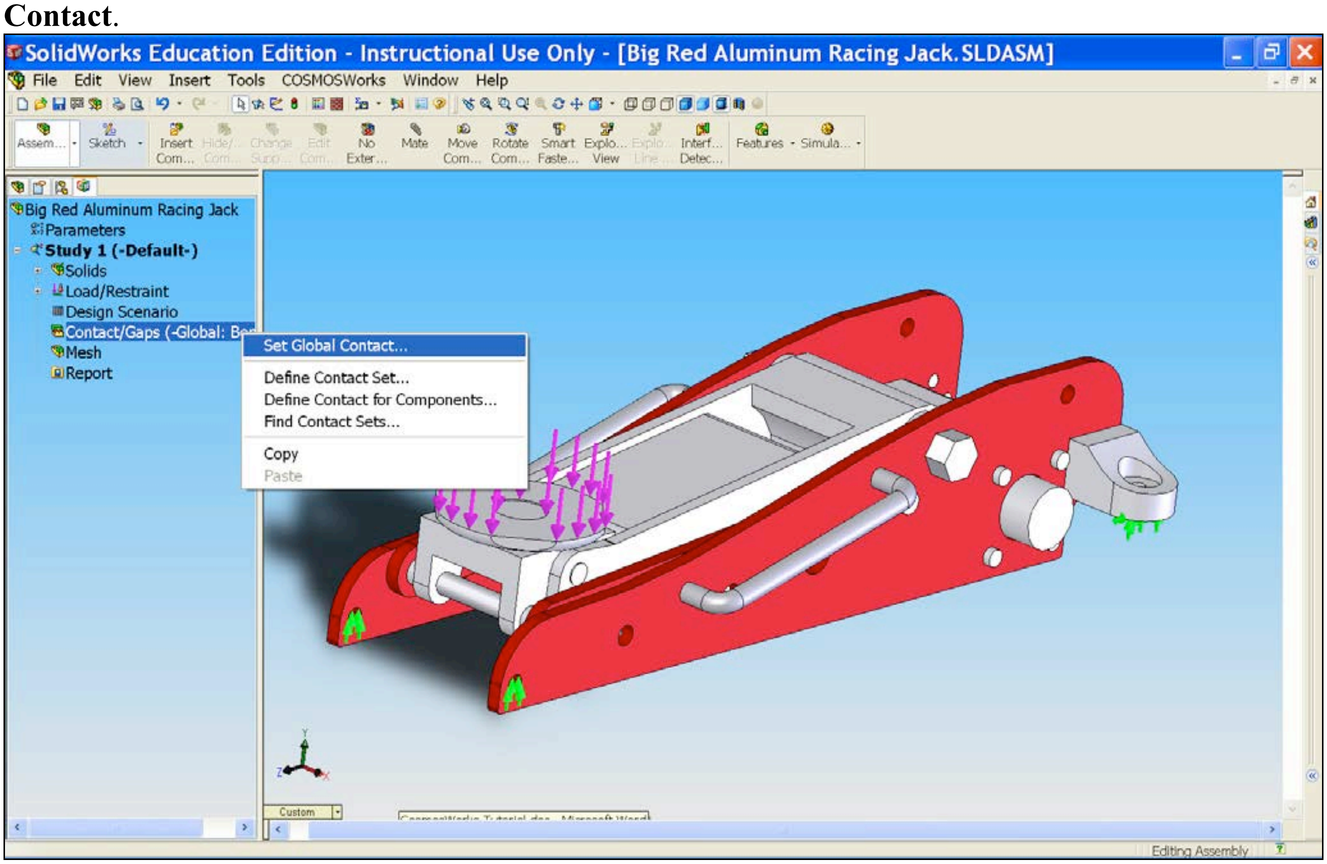
38) Go to **FeatureManager design tree**, right-click **Bottle Jack** and select  **Show**.



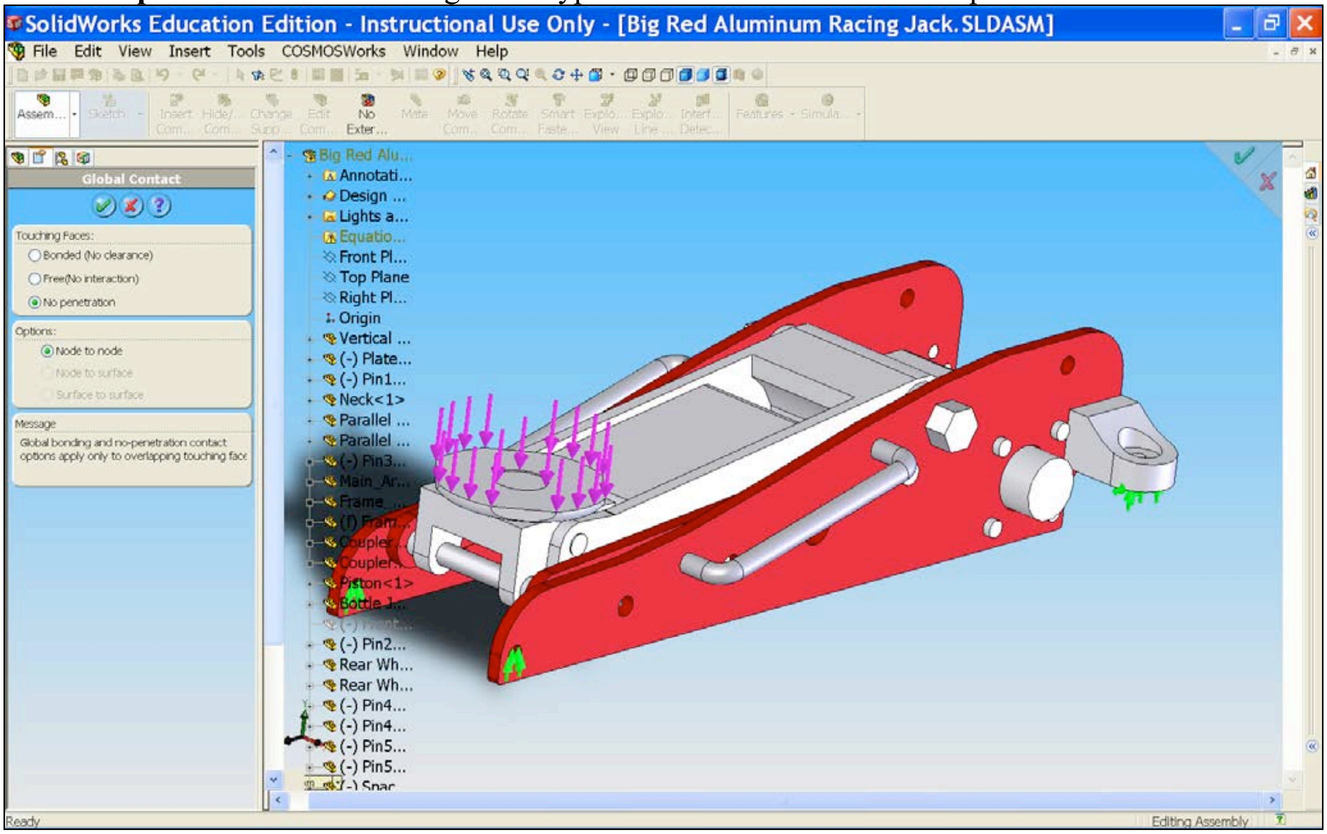
39) The Bottle Jack is now shown back to the assembly.



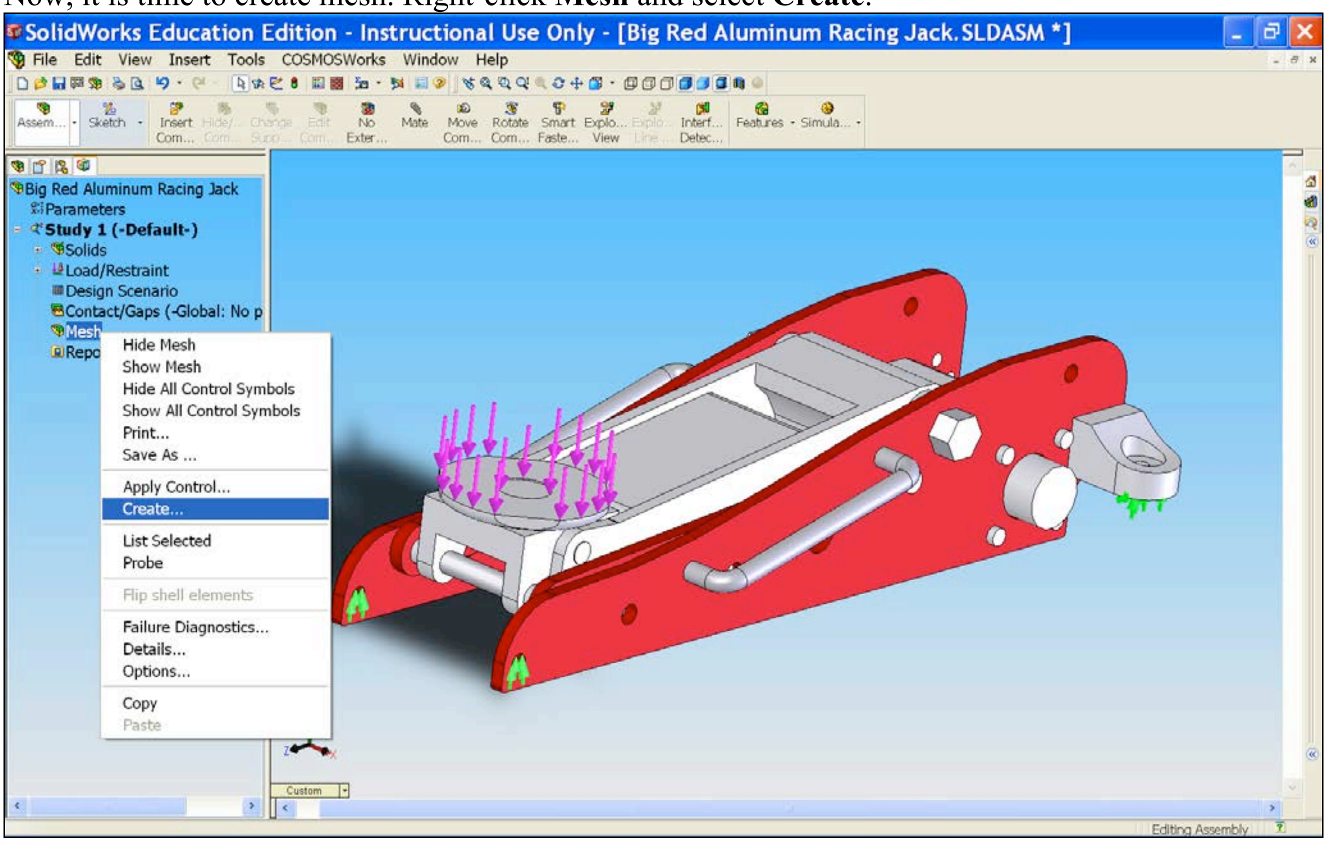
40) Now, we will define a contact type to the assembly. Right-click  **Contact/Gaps** and select **Set Global Contact**.



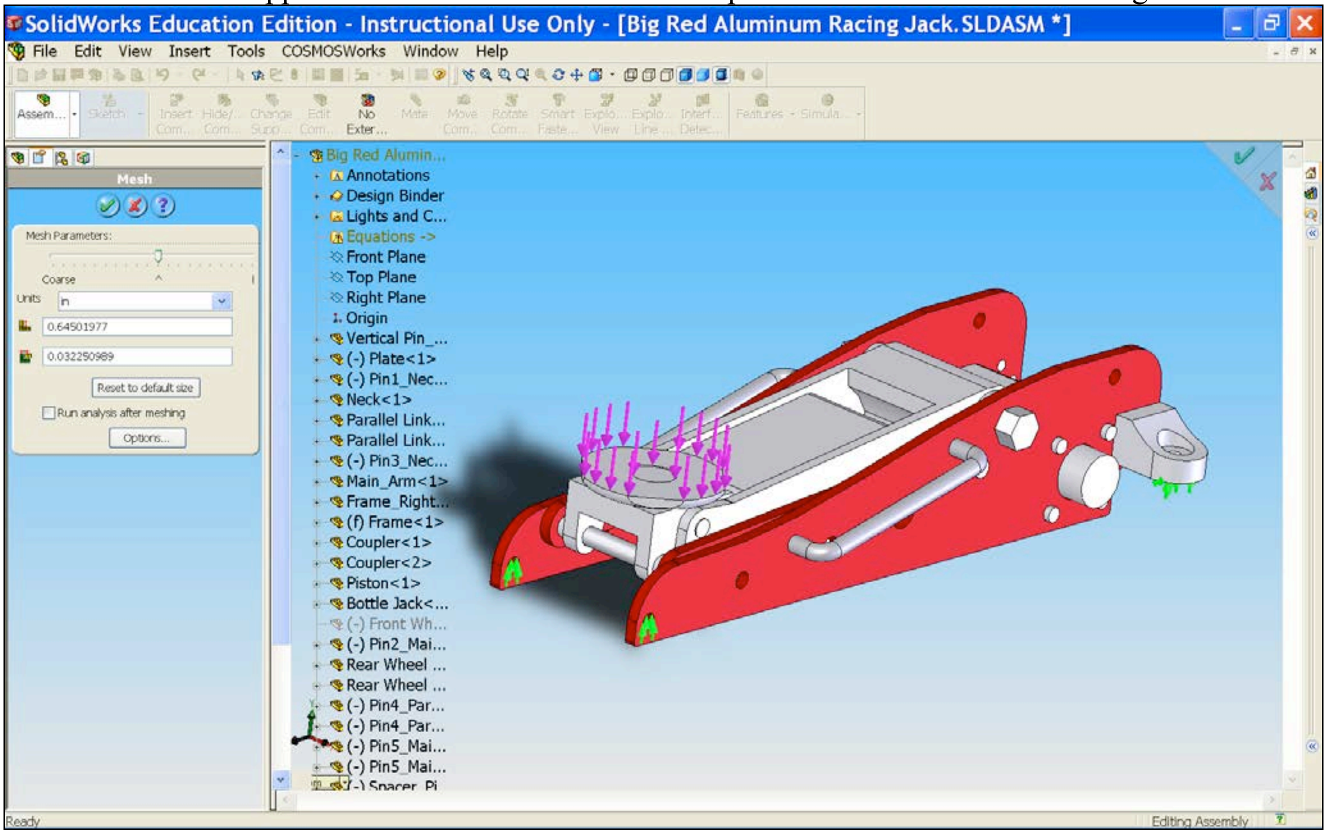
41) Select **No penetration** as Touching Faces type and **Node to node** for the Options. Click OK.



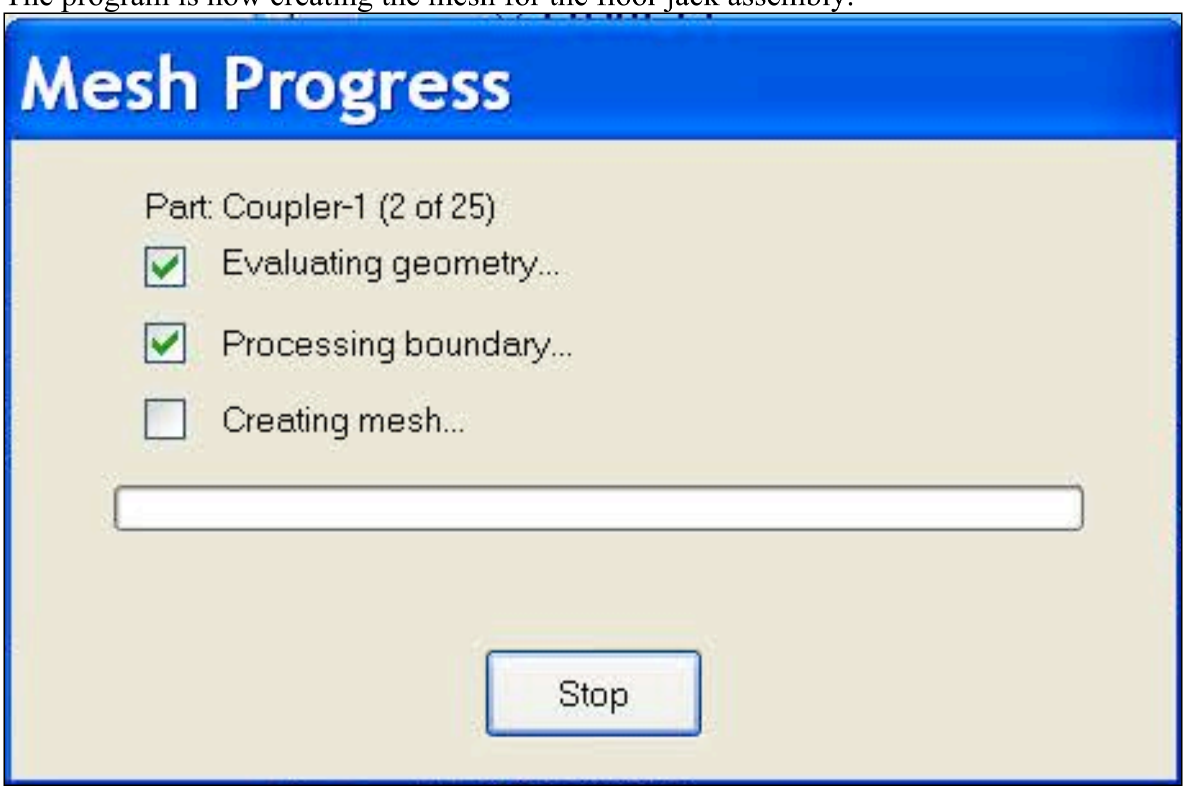
42) Now, it is time to create mesh. Right-click **Mesh** and select **Create**.



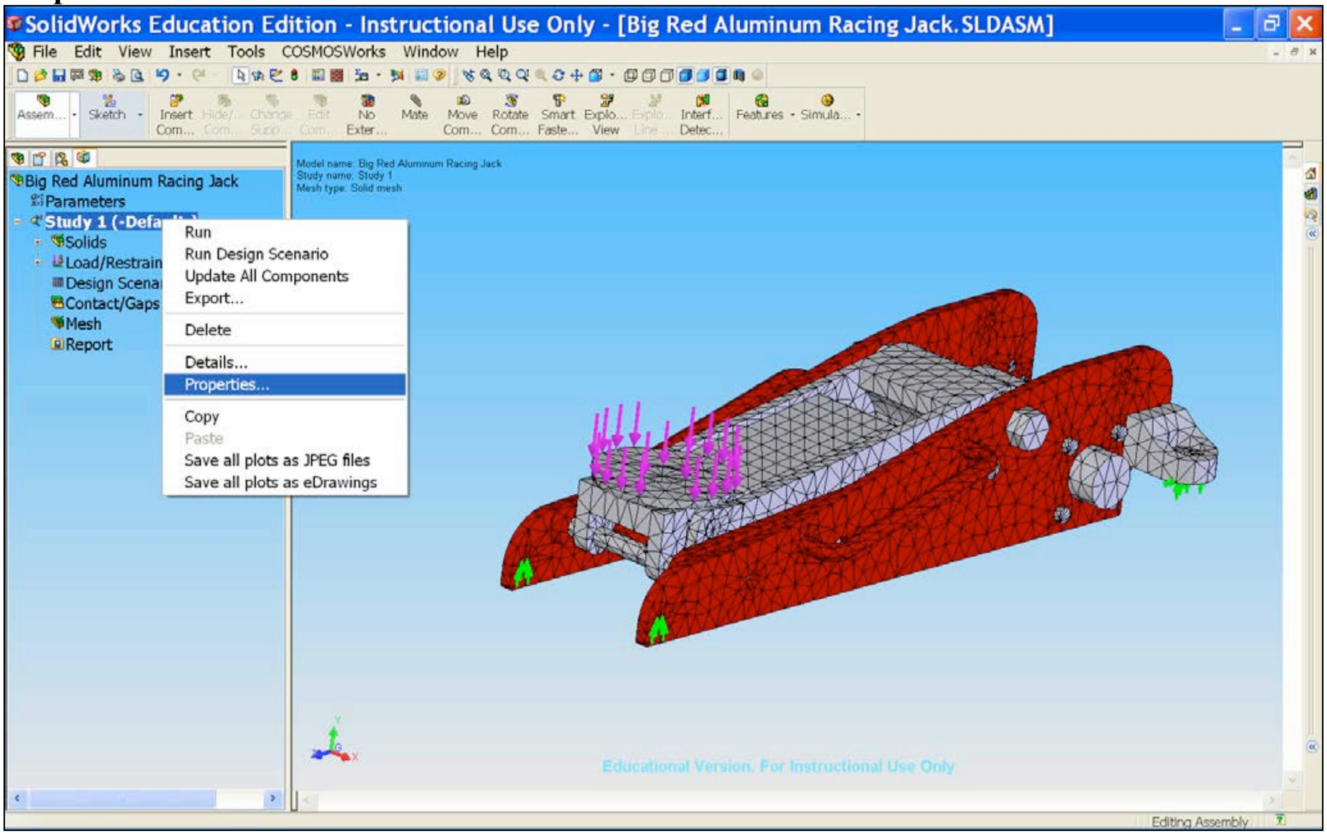
43) The Mesh window appears on the left. Click **OK** to accept the default values for meshing.



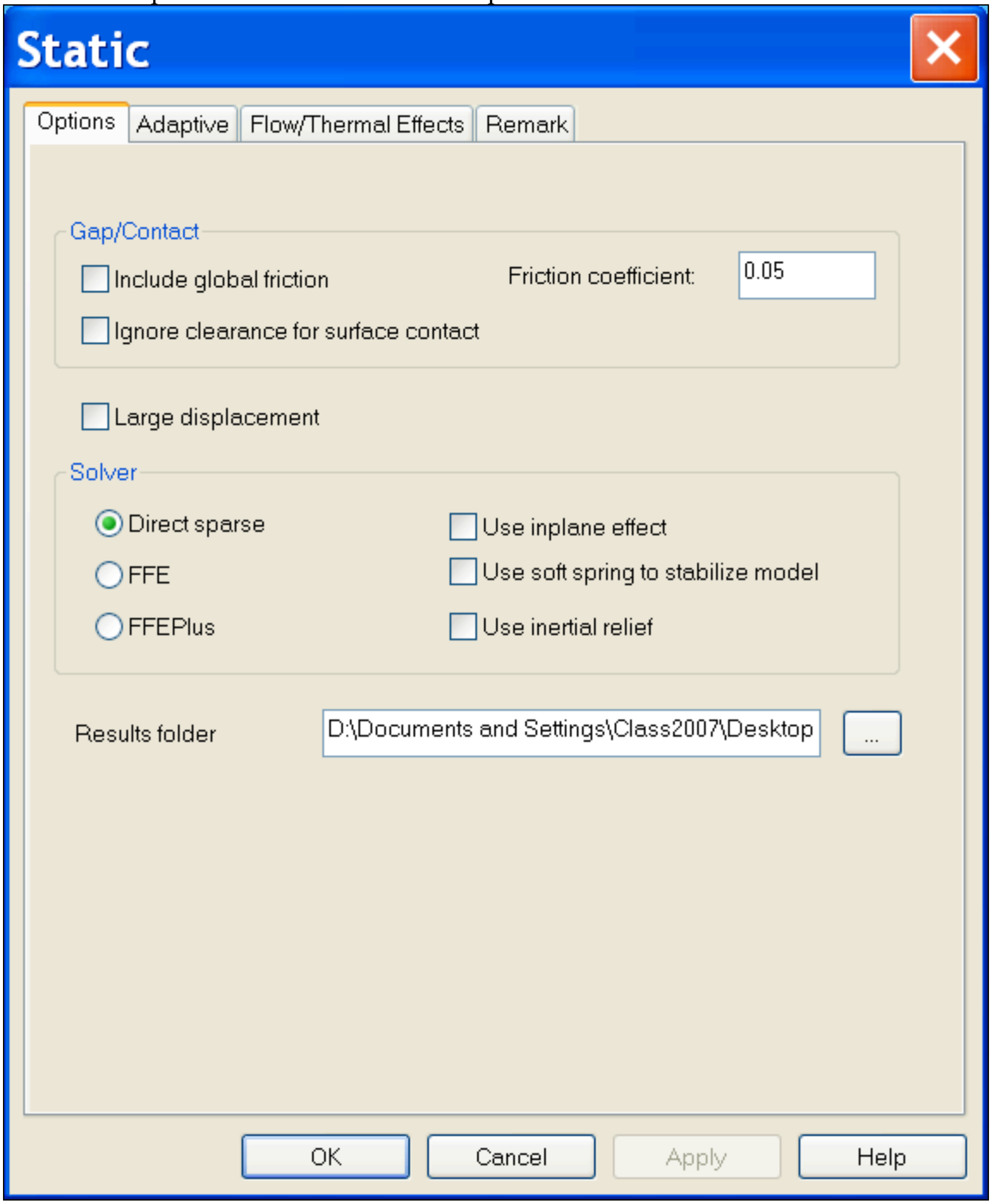
44) The program is now creating the mesh for the floor jack assembly.



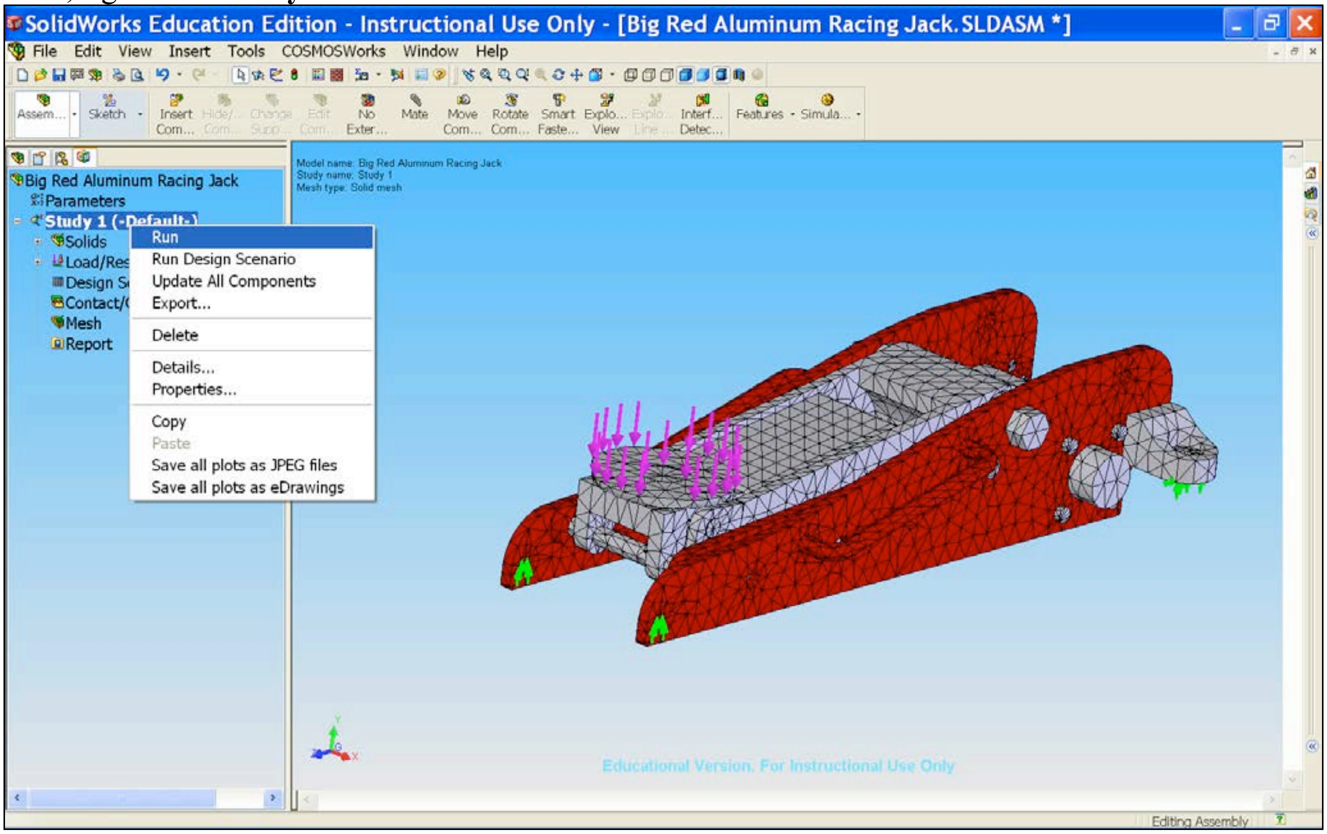
45) Meshing is now complete and it is time to run the COSMOSWorks analysis. Right-click **Study 1** and select **Properties**.



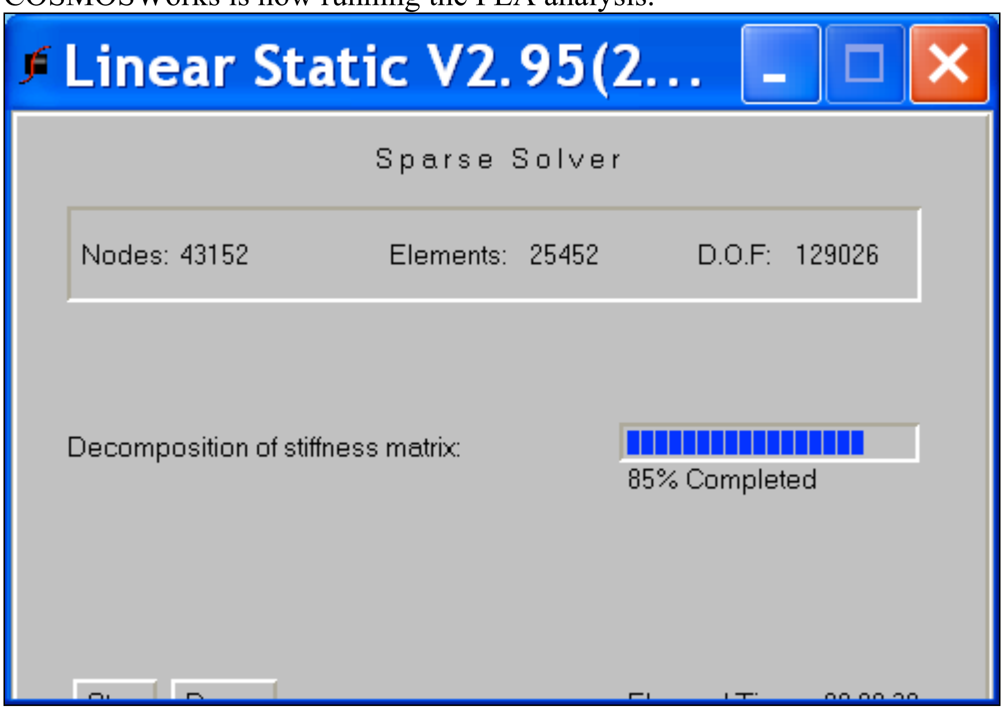
46) Select **Direct sparse** as a solver and click **OK**. Using Direct sparse as a solver will minimize the time to run the analysis and it also gives more accurate results. However, the downside of using this solver is that it uses too much RAM space which will cause the computer to run slow.



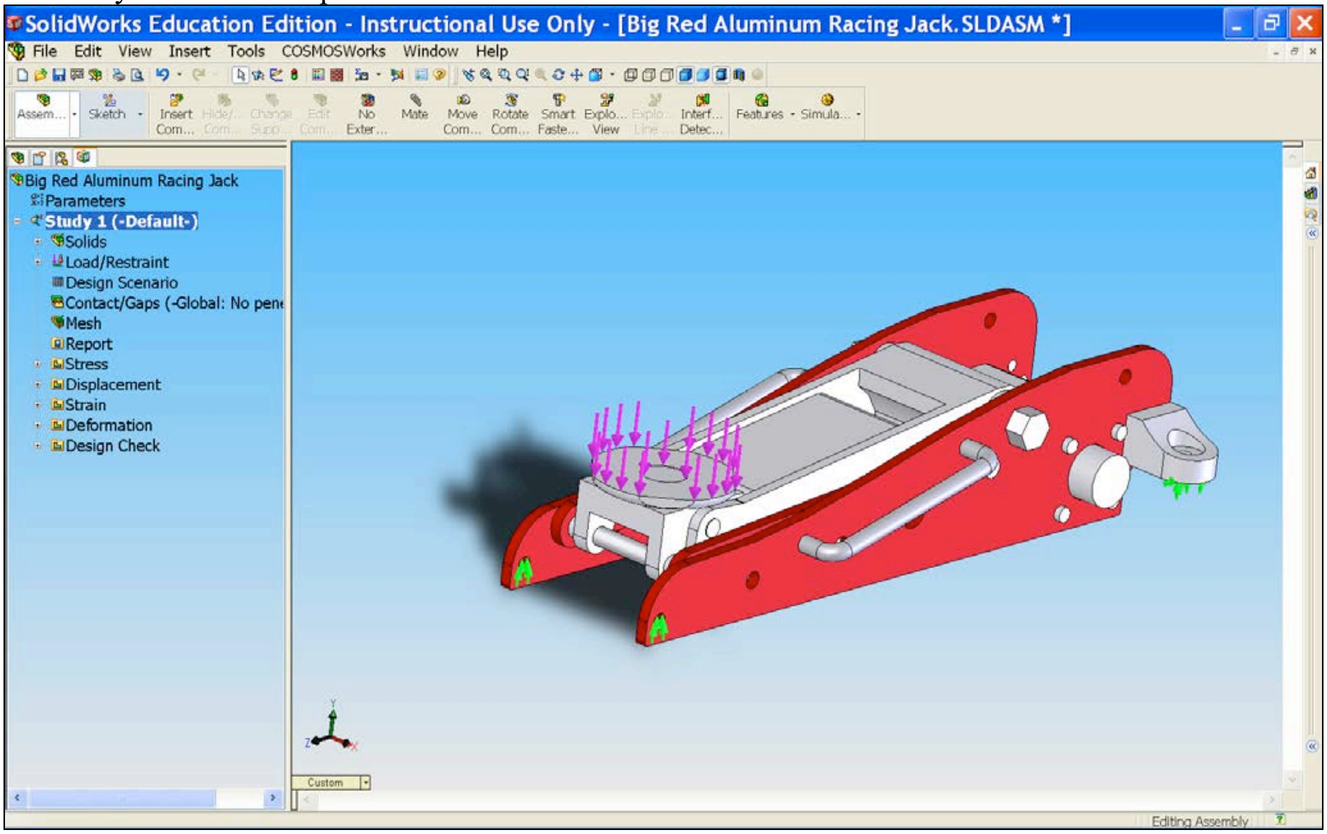
47) Now, right-click **Study 1** and select **Run**.



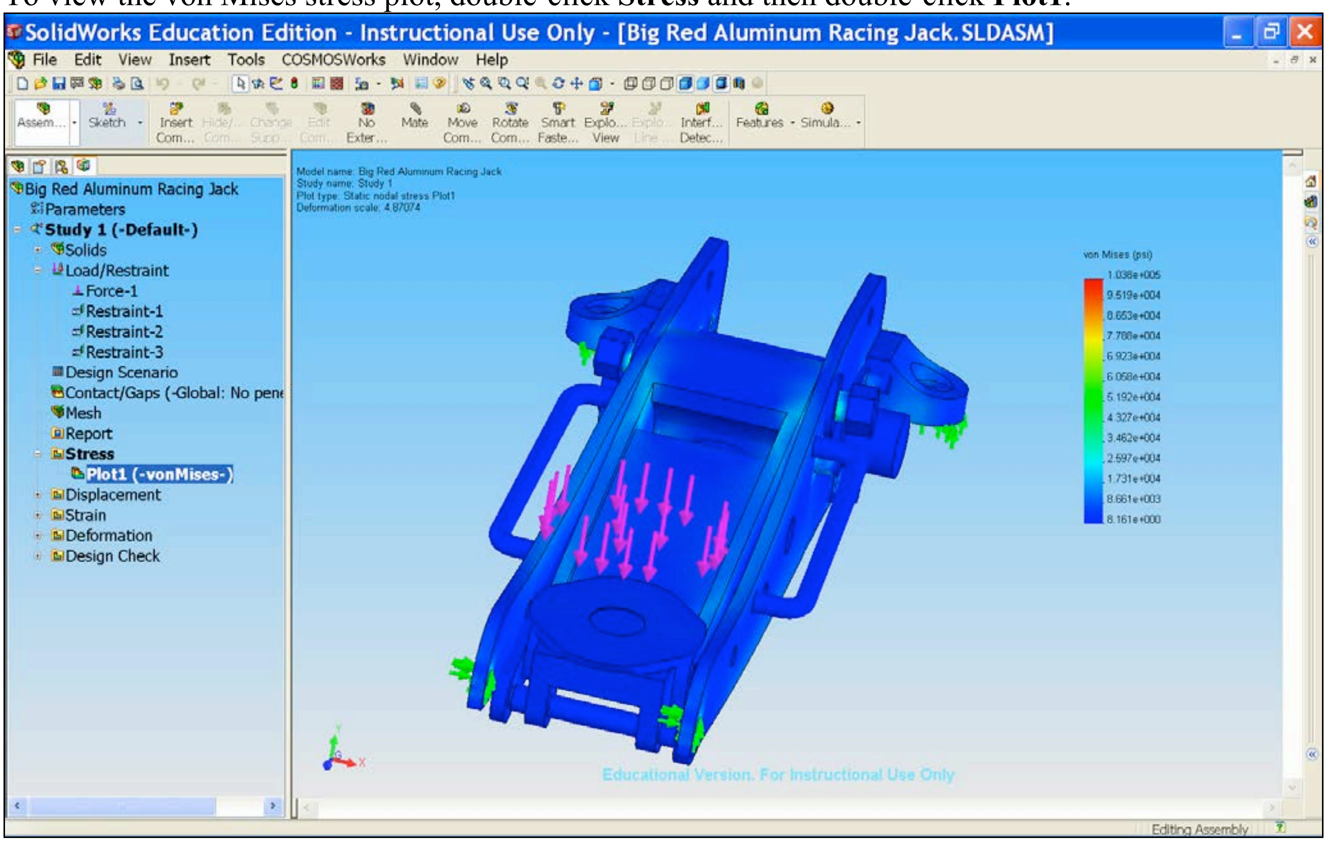
48) COSMOSWorks is now running the FEA analysis.



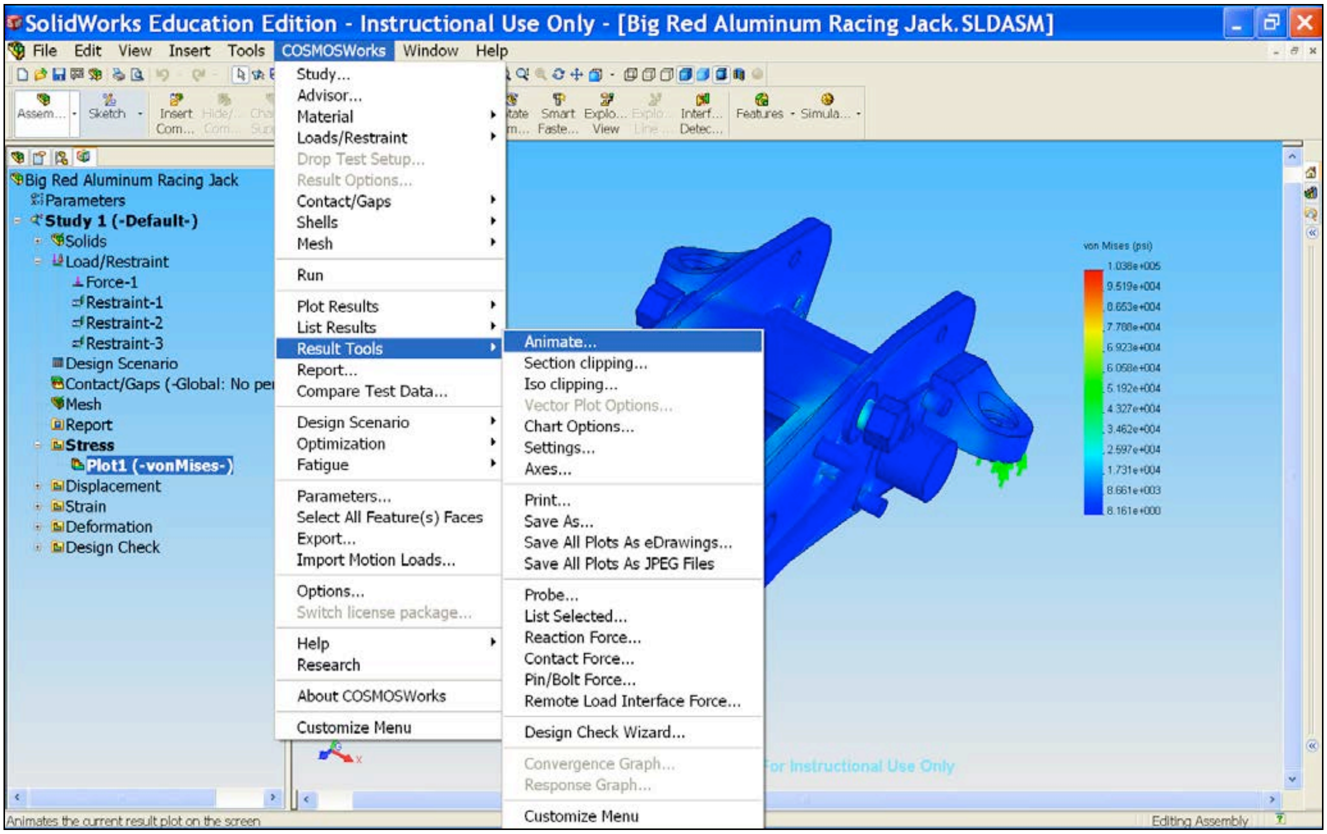
49) The analysis is now complete.



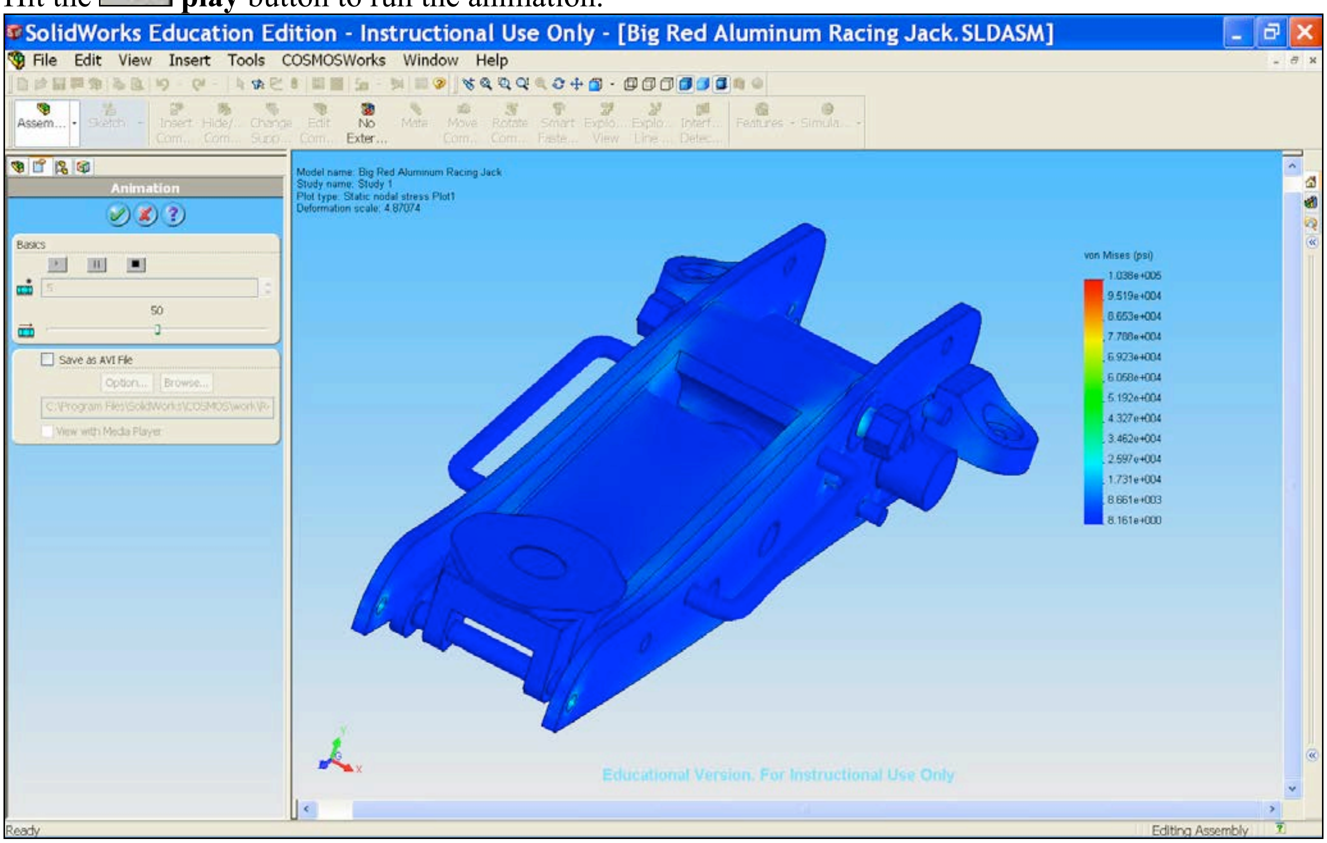
50) To view the von Mises stress plot, double-click **Stress** and then double-click **Plot1**.



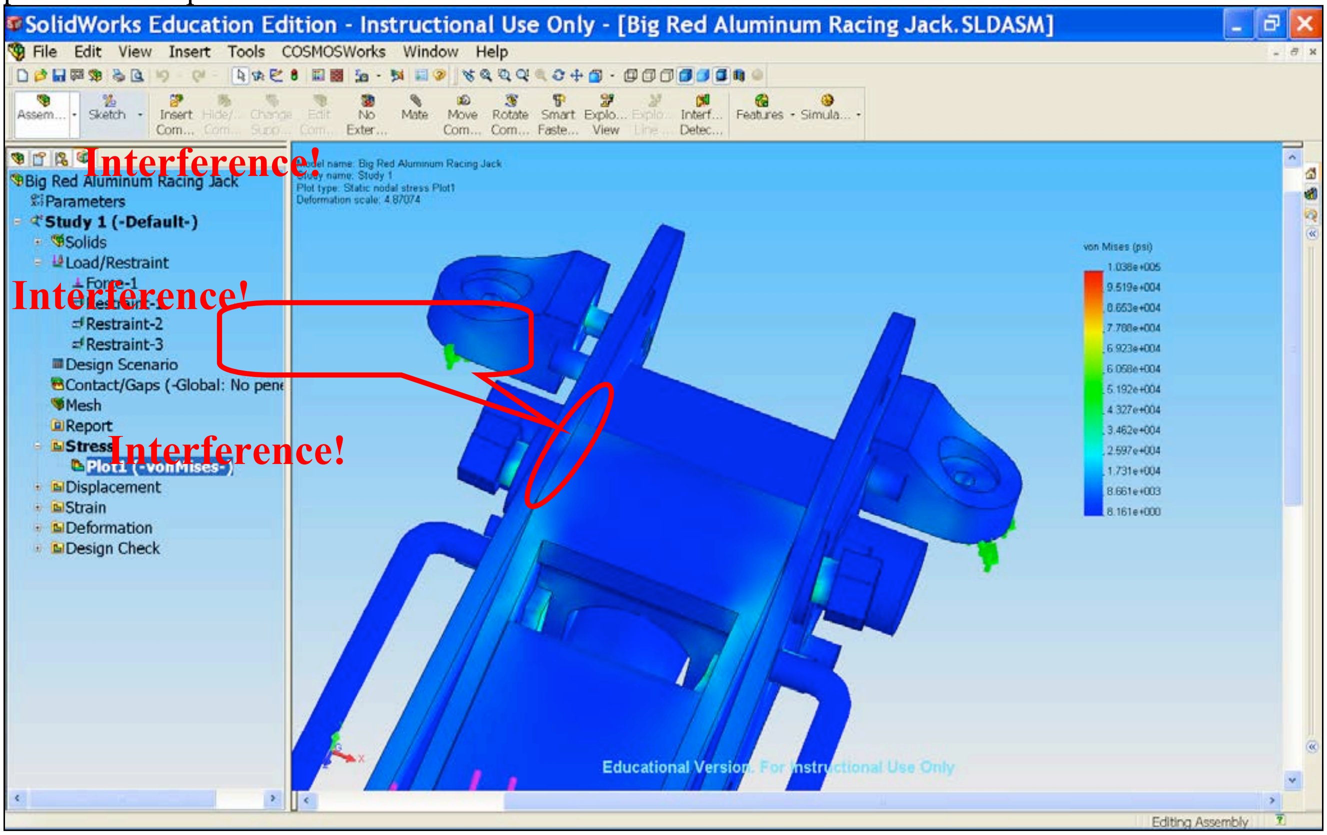
51) To better understand how the parts deform under loading, we can use an animation function to show us the stress and deformation of the assembly over a period of time. Go to **COSMOSWorks** menu on top, then go to **Result Tools** and select **Animate**.



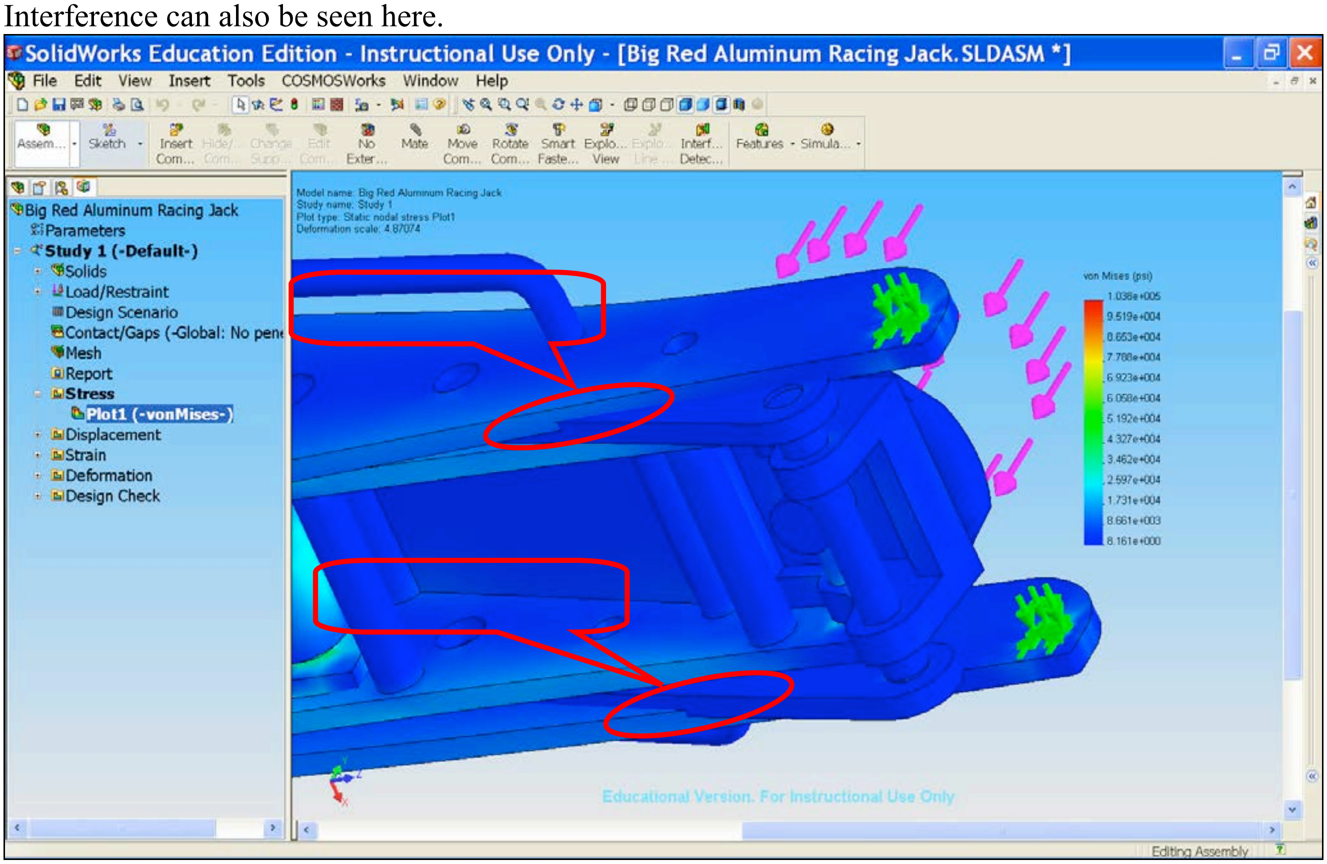
52) Hit the  **play** button to run the animation.



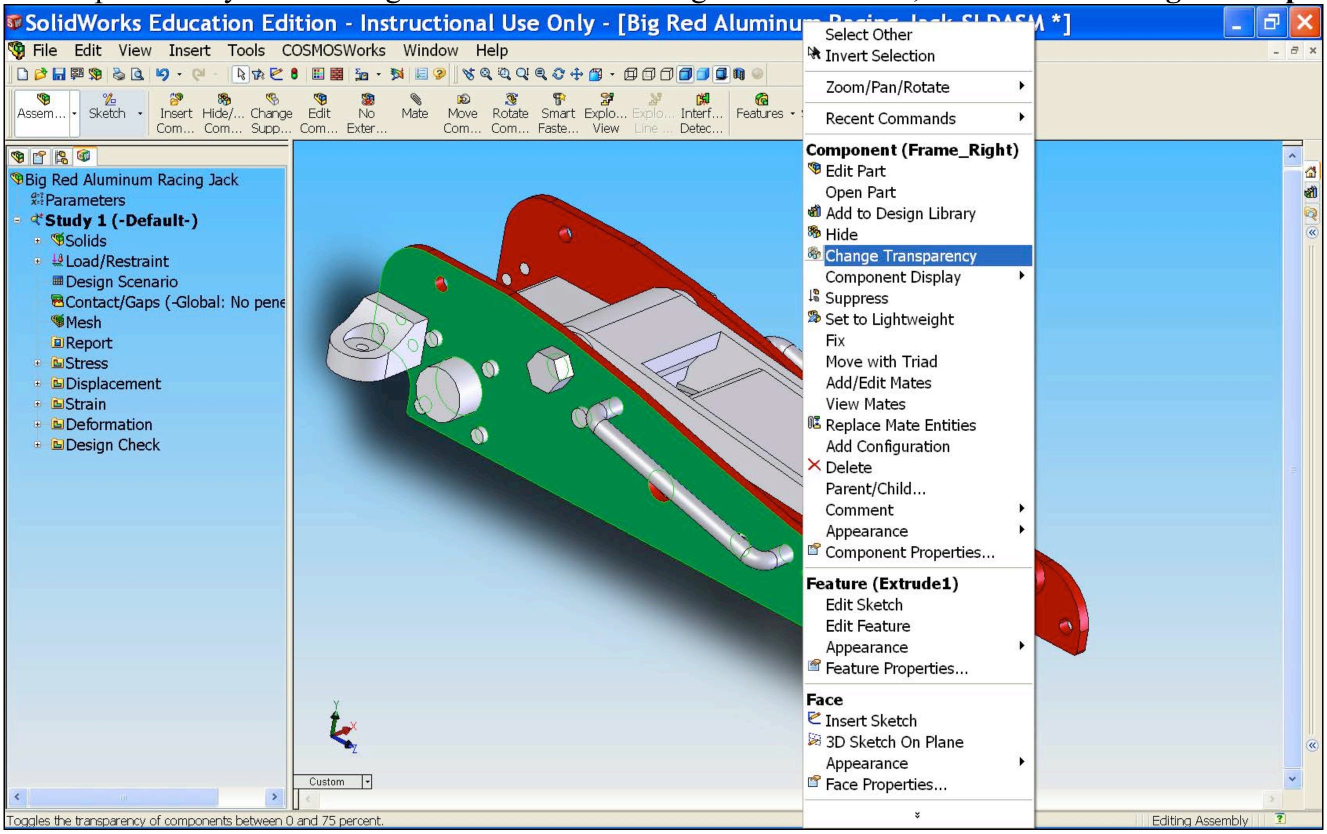
53) Note that some components actually intersect each other. This is not desirable because in real life, parts interference does not occur. Therefore, accurate results will not be obtained if the interference is present. The picture below points out the interferences.



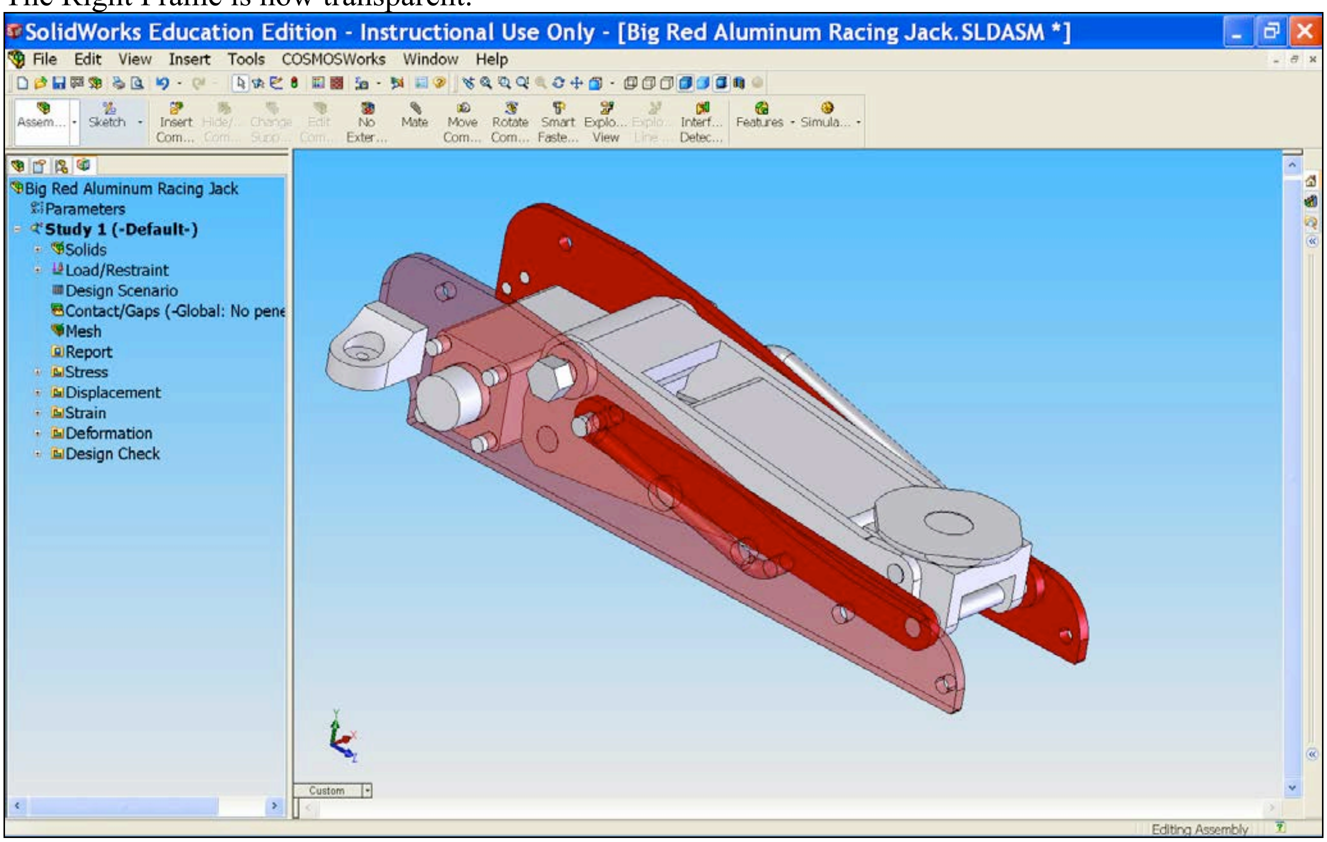
54) Interference can also be seen here.



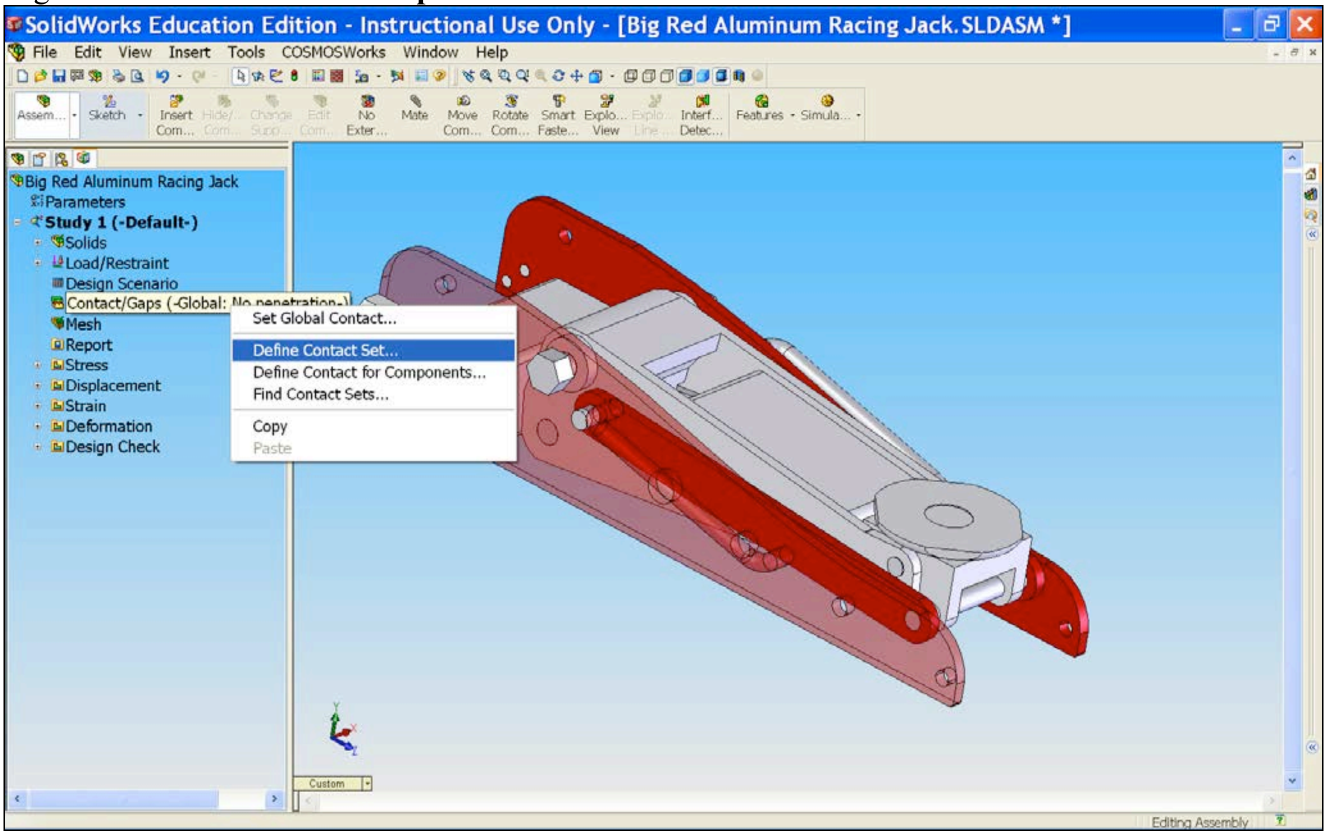
55) To fix the interference problem, we need to further define the contact or gaps for the assembly. First, put the mouse pointer anywhere on right frame surface and right-click it. Then, select  **Change Transparency**.



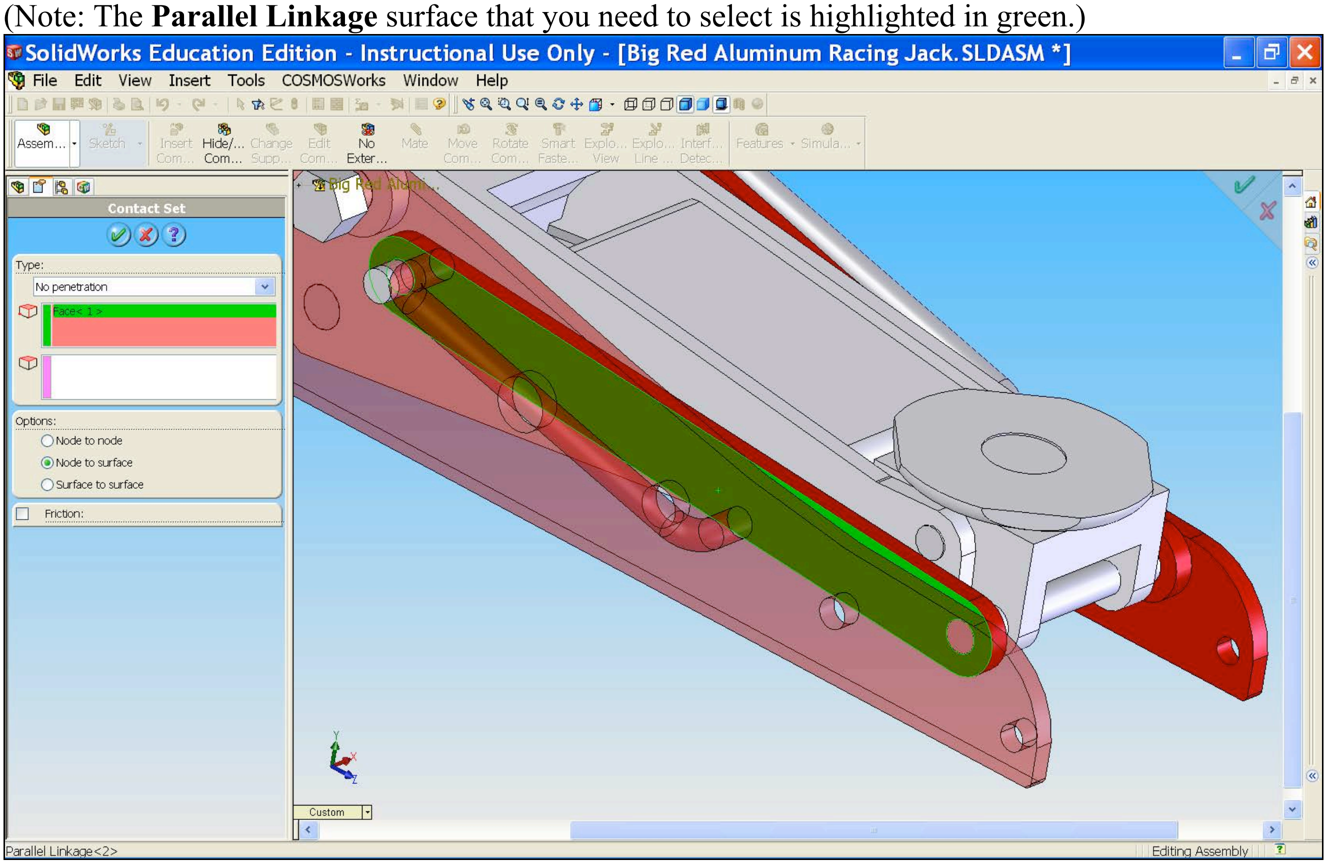
56) The Right Frame is now transparent.



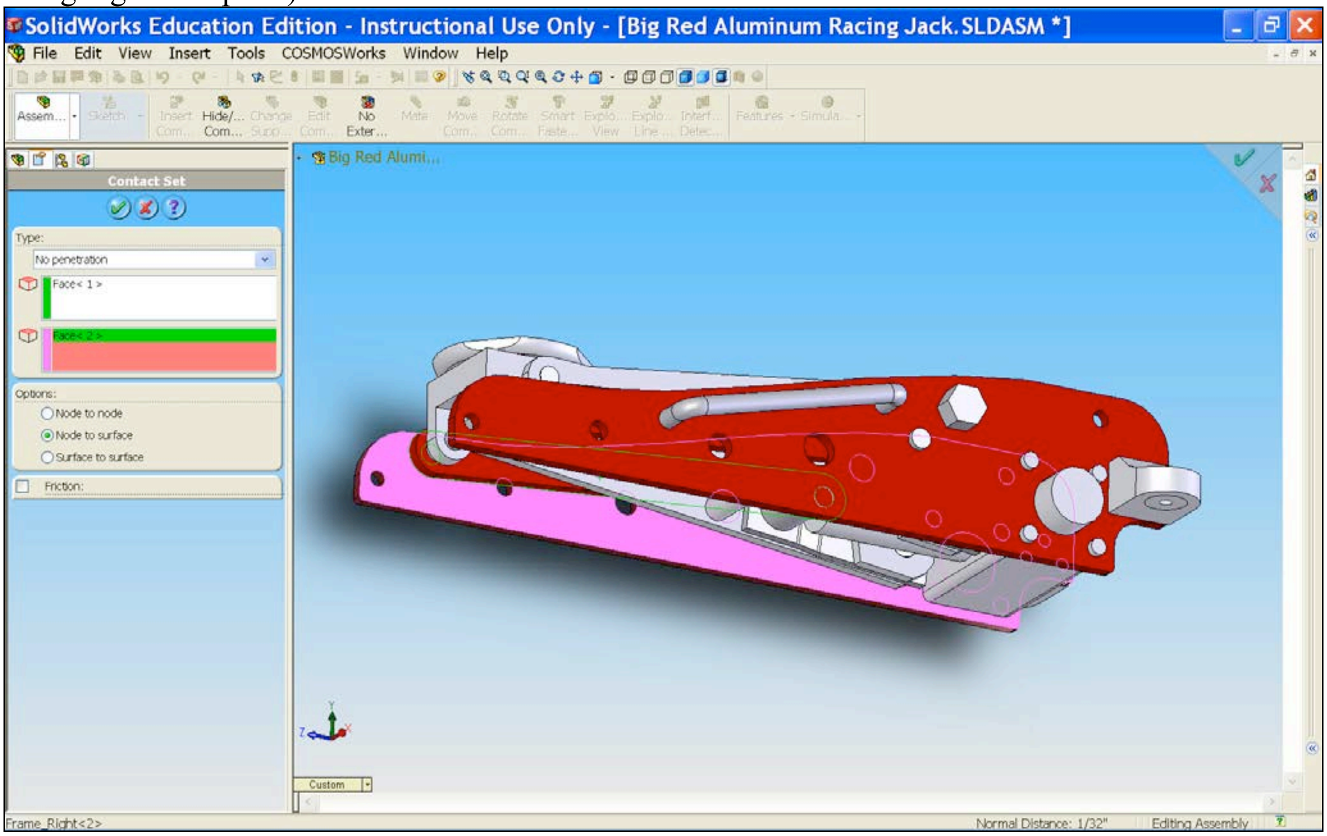
57) Right-click  **Contact/Gaps** and select **Define Contact Set**.



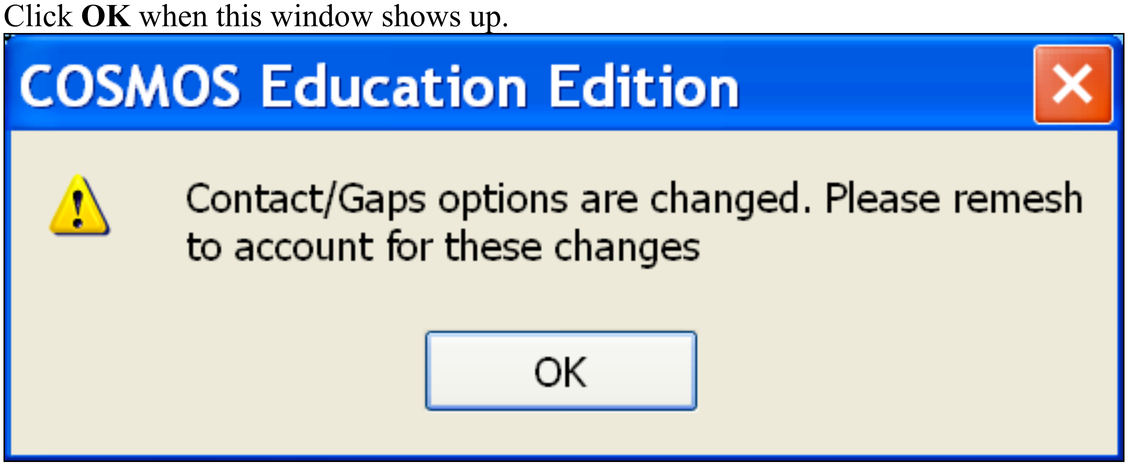
58) For  **Faces, Edges, Vertices for Source,** select the face of **Parallel Linkage** as shown in picture below. (Note: The **Parallel Linkage** surface that you need to select is highlighted in green.)



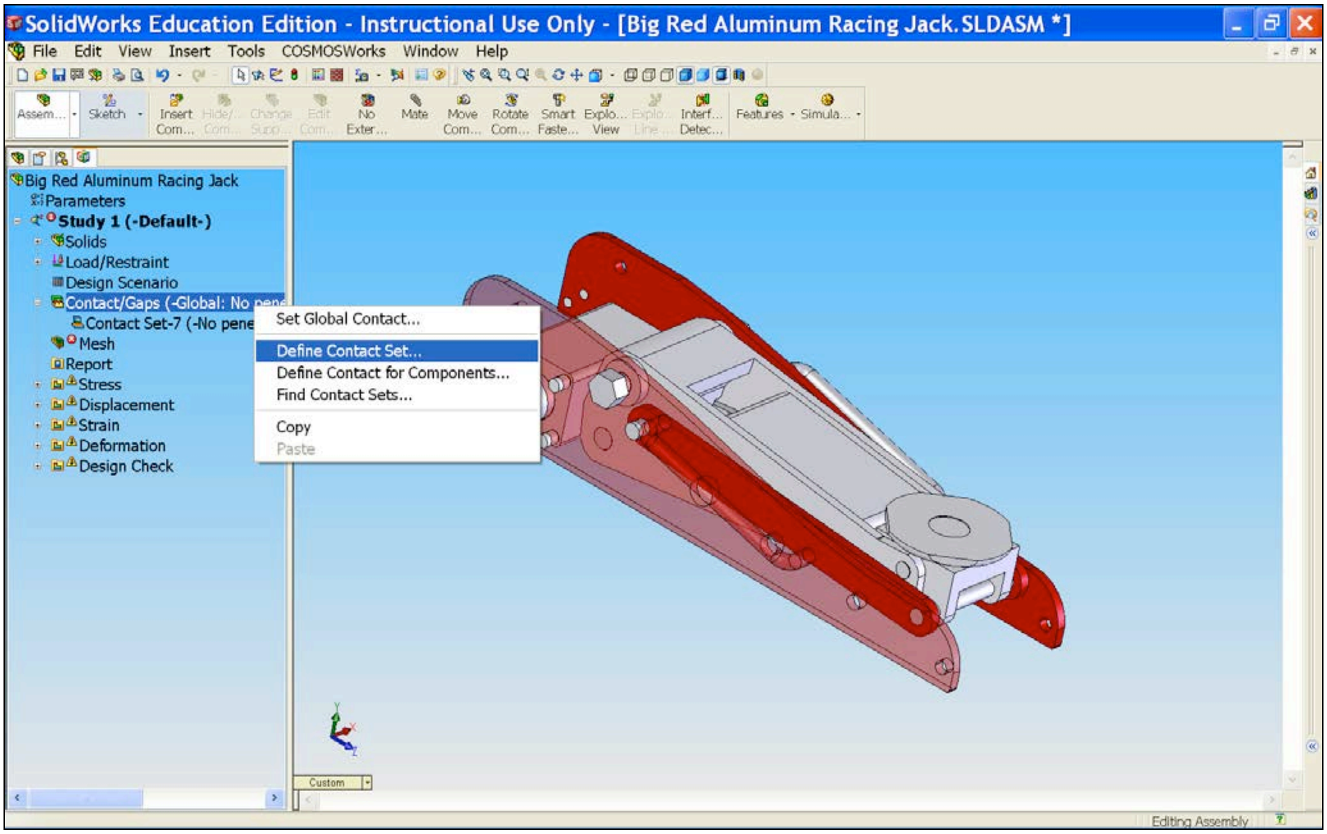
59) For  **Faces for Target**, select the surface of the **Right Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Right Frame** surface that you need to select is highlighted in pink.)



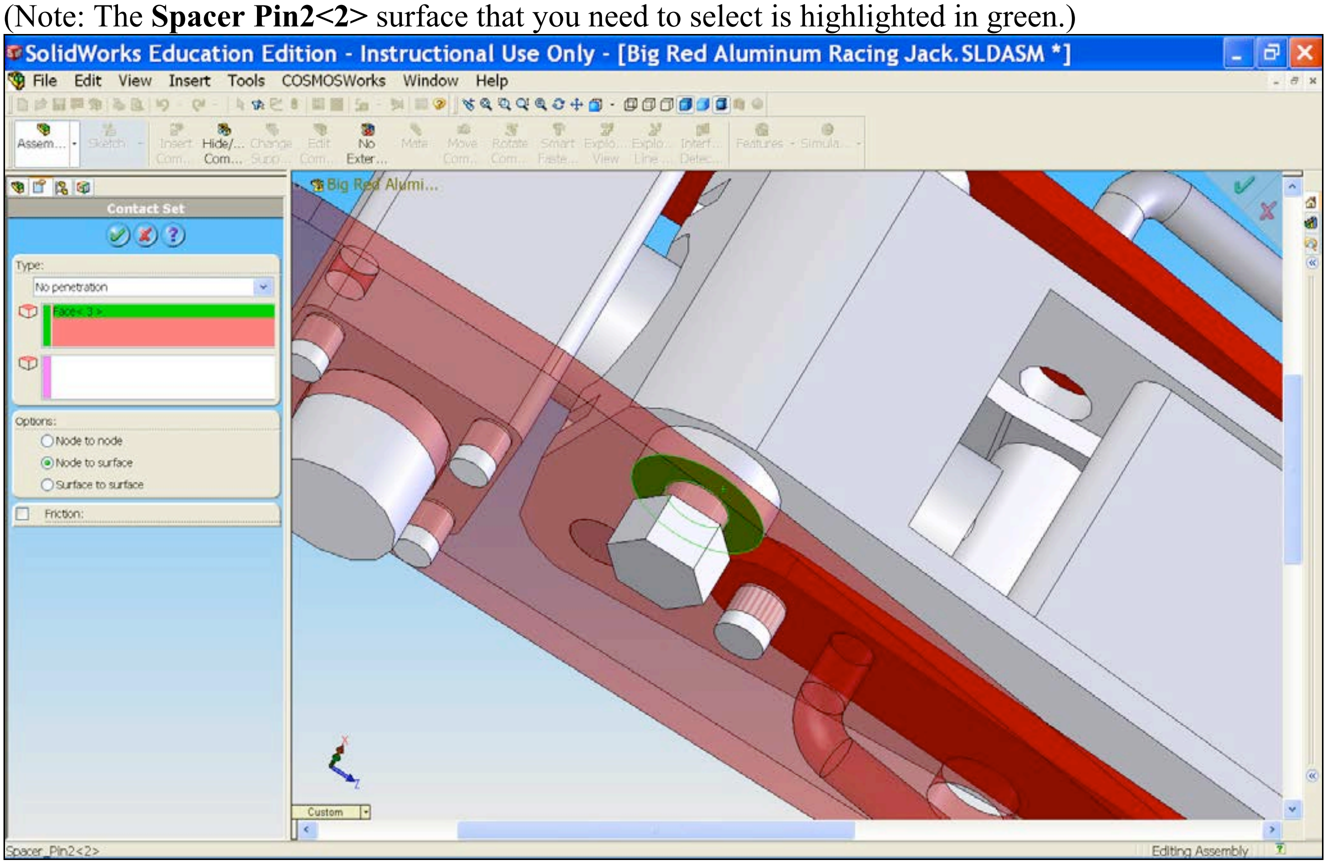
60) Click **OK** when this window shows up.



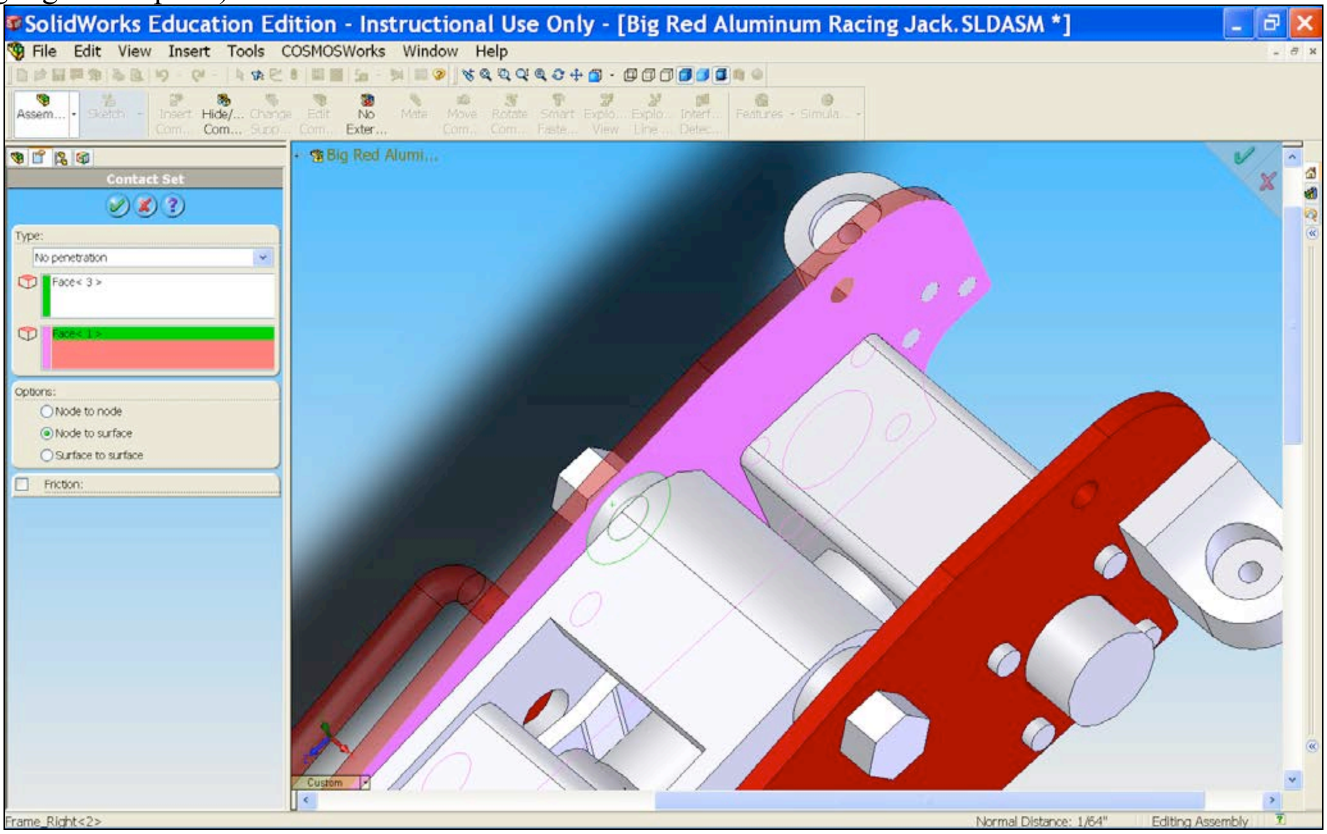
61) Now, we will continue defining the contact sets for the other components. Right-click  **Contact/Gaps** and select **Define Contact Set**.



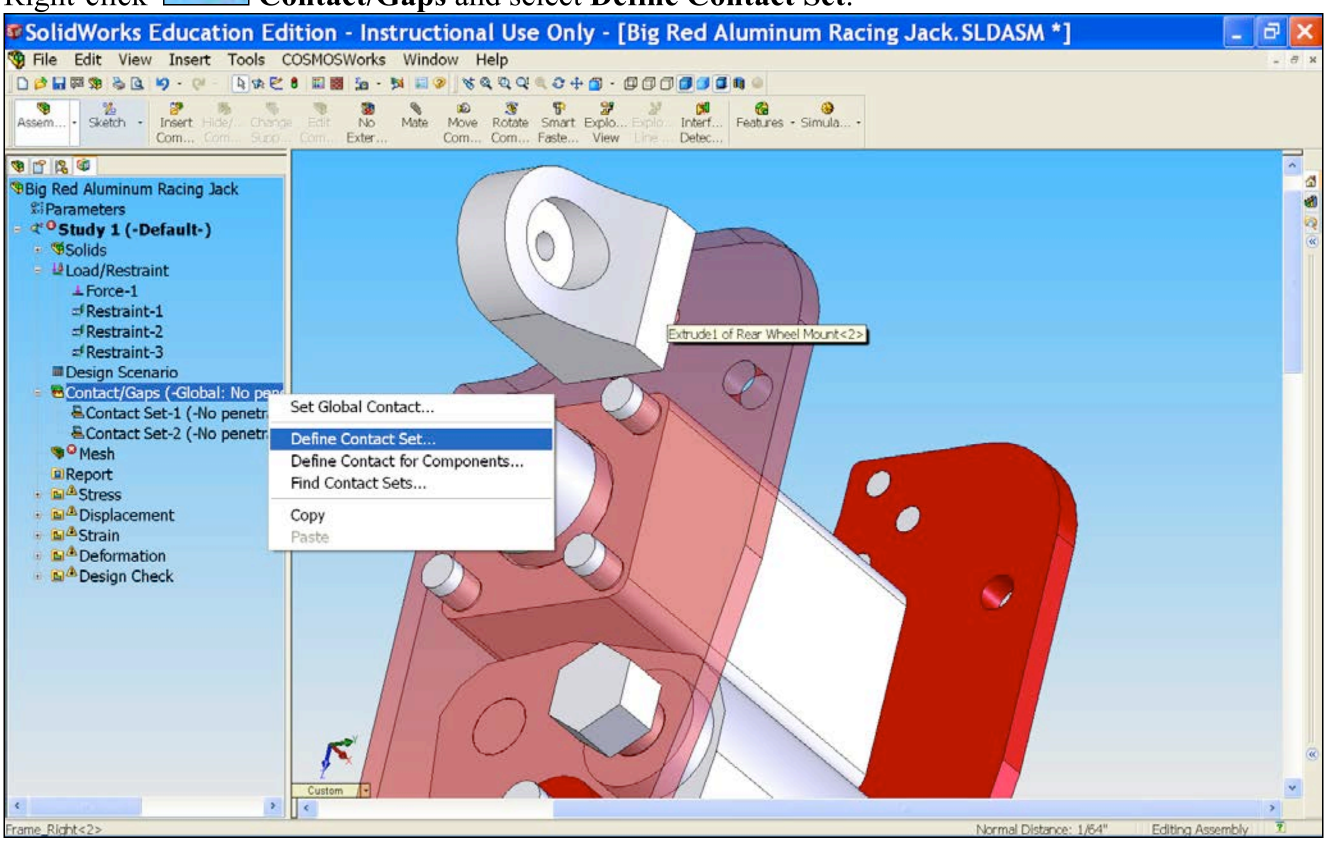
62) For  **Faces, Edges, Vertices for Source,** select the face of **Spacer Pin2<2>** as shown in picture below. (Note: The **Spacer Pin2<2>** surface that you need to select is highlighted in green.)

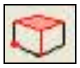


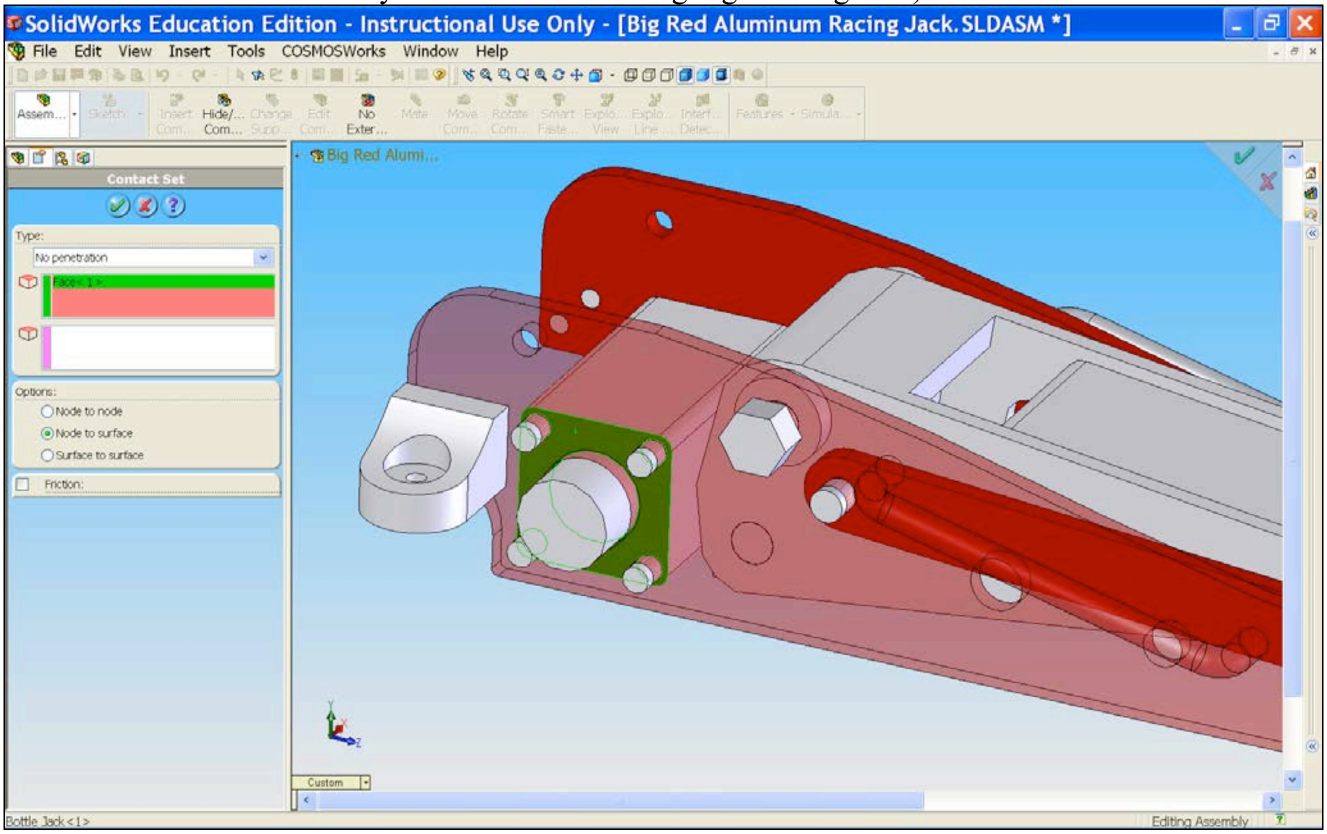
63) For  **Faces for Target**, select the surface of the **Right Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Right Frame** surface that you need to select is highlighted in pink.)



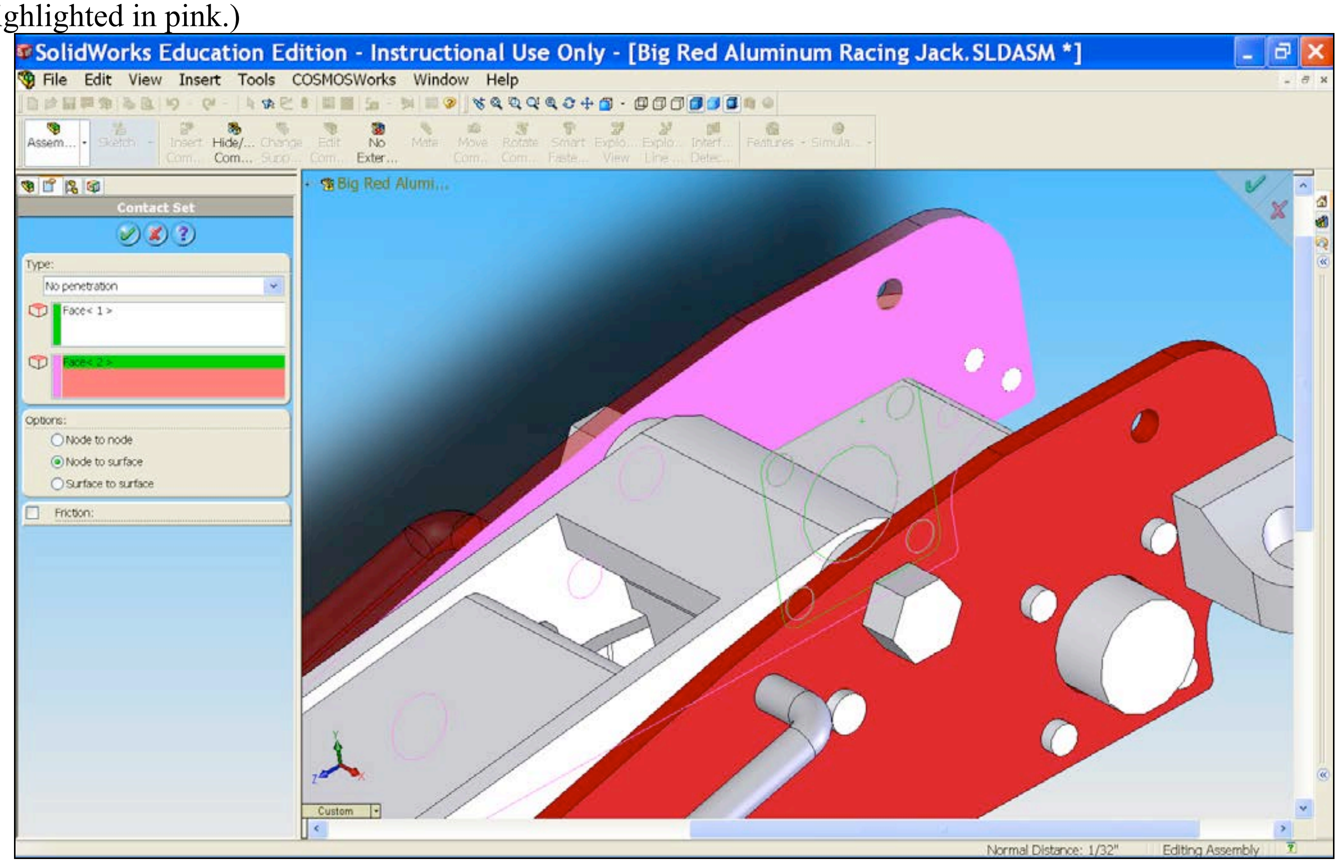
64) Right-click  **Contact/Gaps** and select **Define Contact Set**.



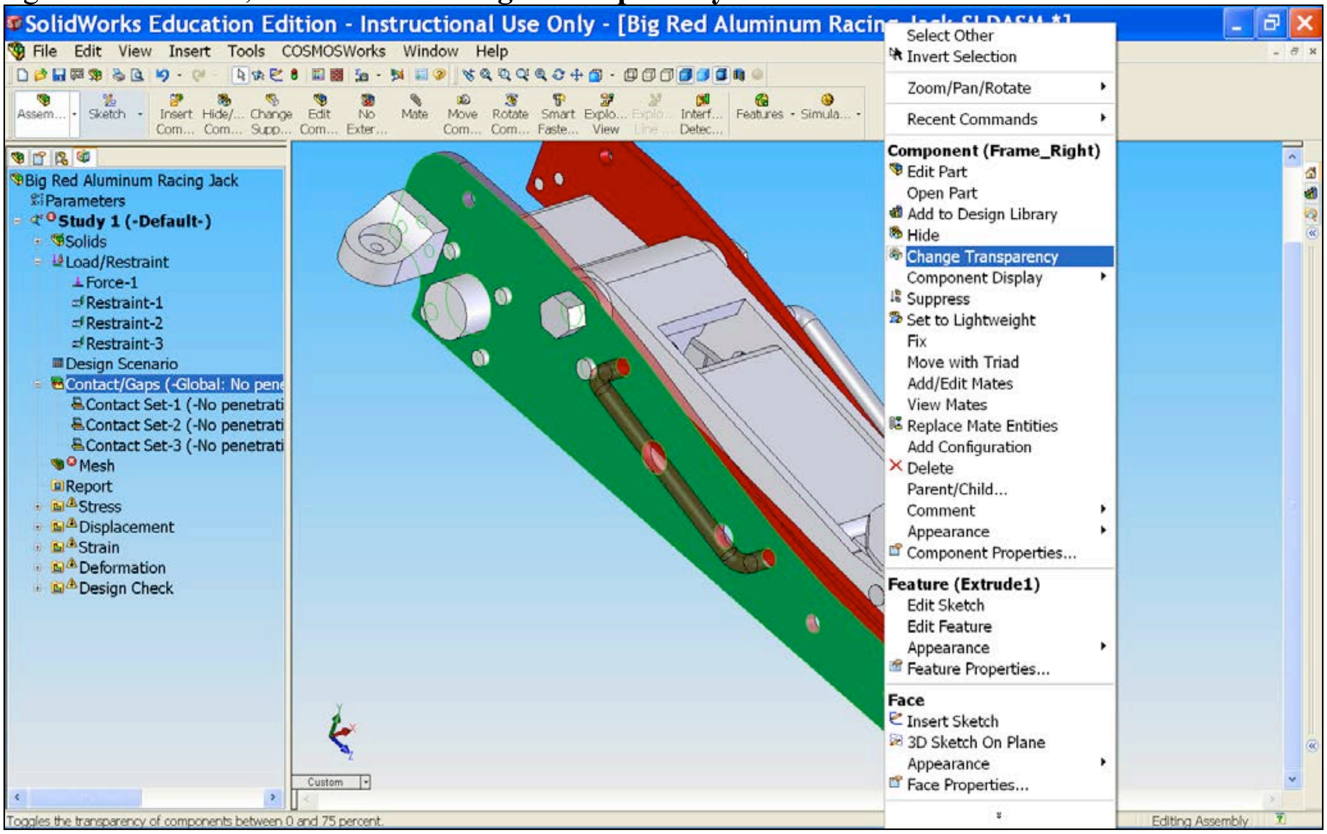
65) For  **Faces, Edges, Vertices for Source,** select the face of **Bottle Jack** as shown in picture below. (Note: The **Bottle Jack** surface that you need to select is highlighted in green.)



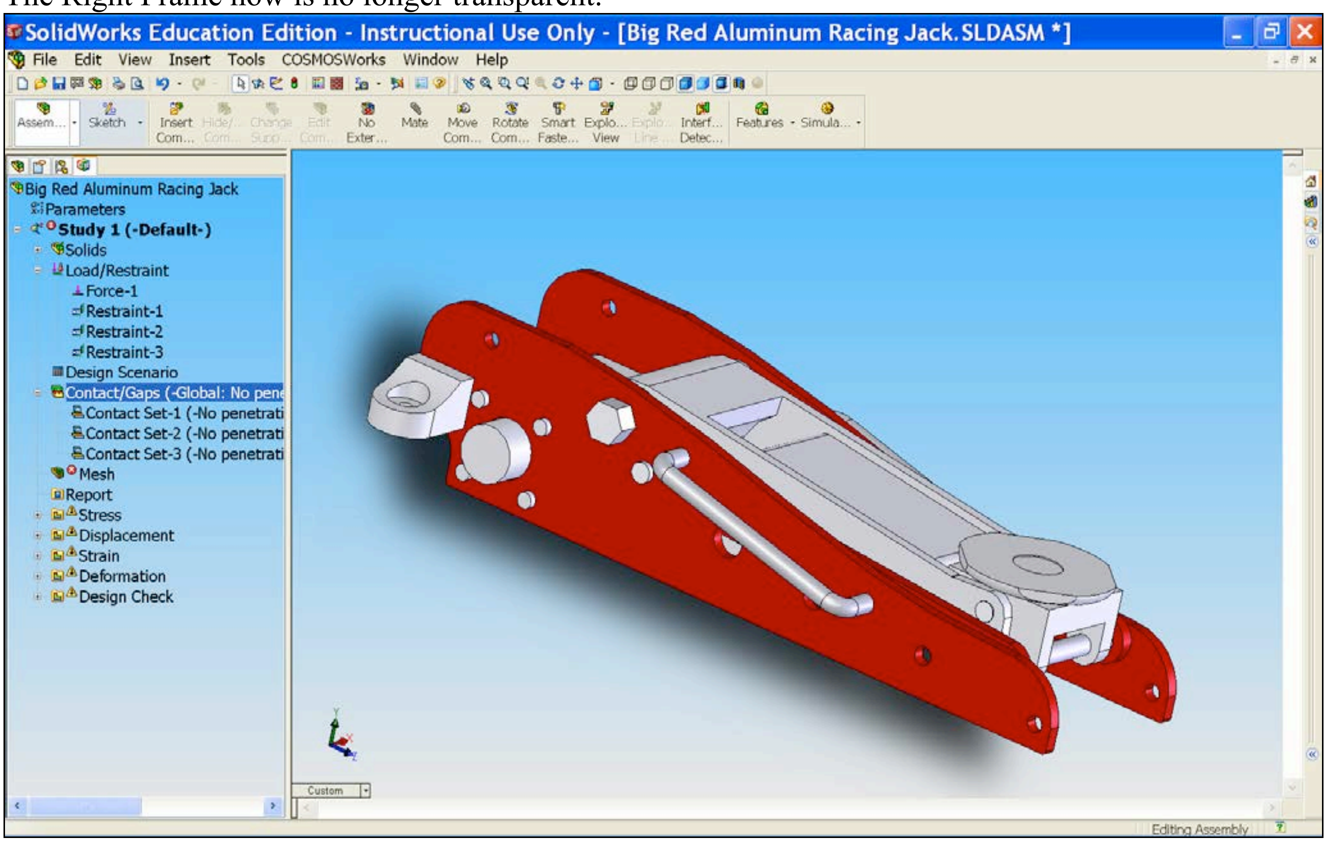
66) For  **Faces for Target**, select the surface of the **Right Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Right Frame** surface that you need to select is highlighted in pink.)



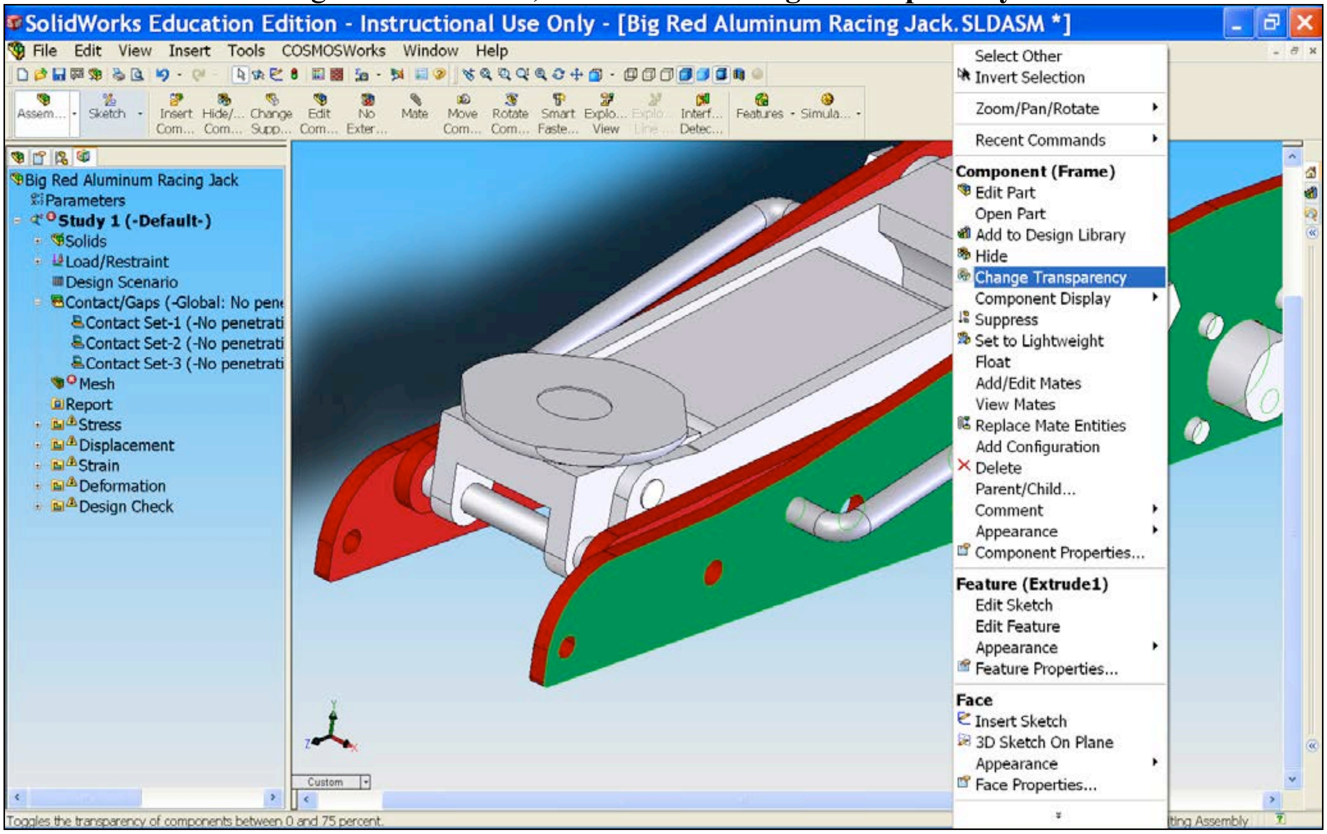
67) We have completed defining gaps to avoid interference of the parts with the right frame. We can now change the transparency of the right frame back to normal. Put the mouse pointer anywhere on right frame surface and right-click it. Then, select  **Change Transparency**.



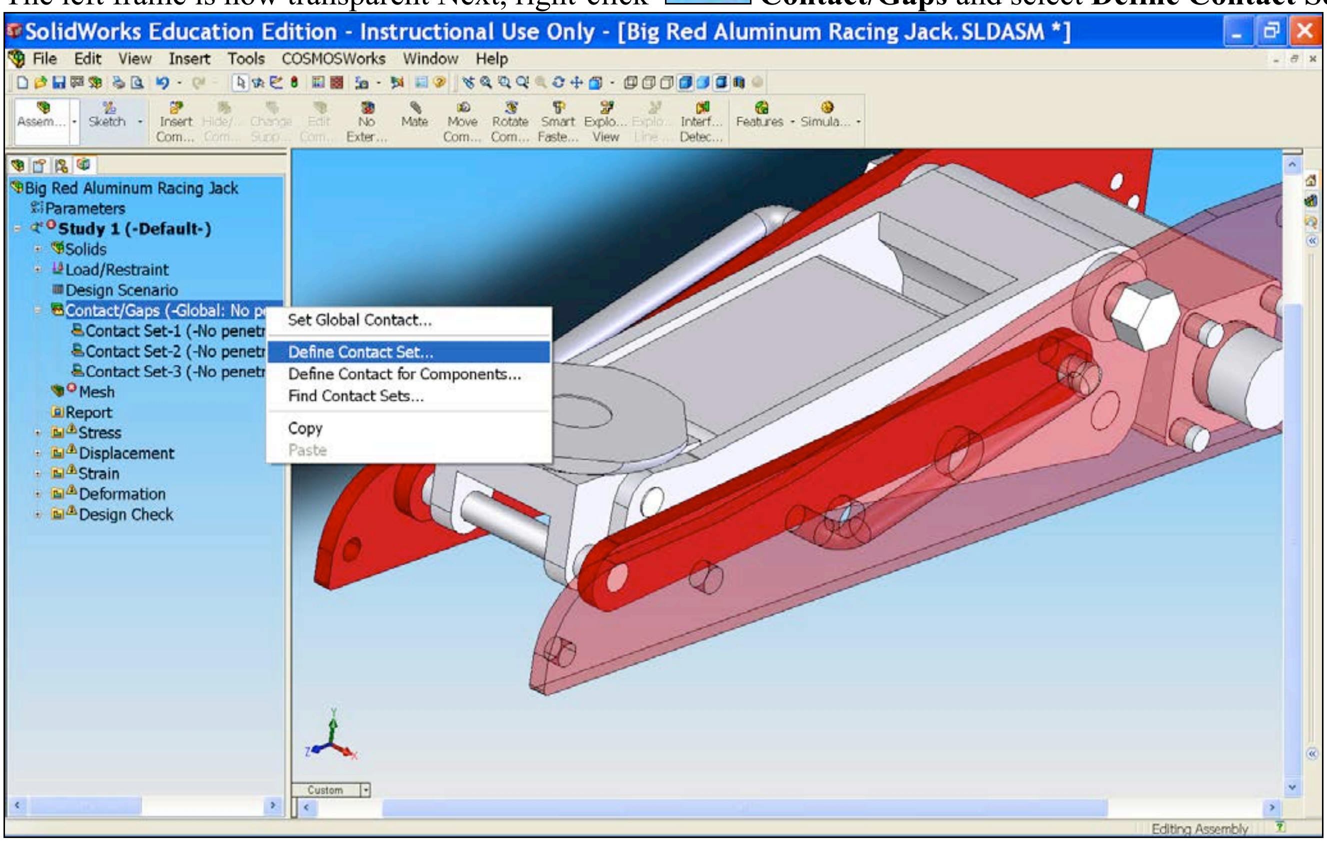
68) The Right Frame now is no longer transparent.

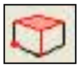


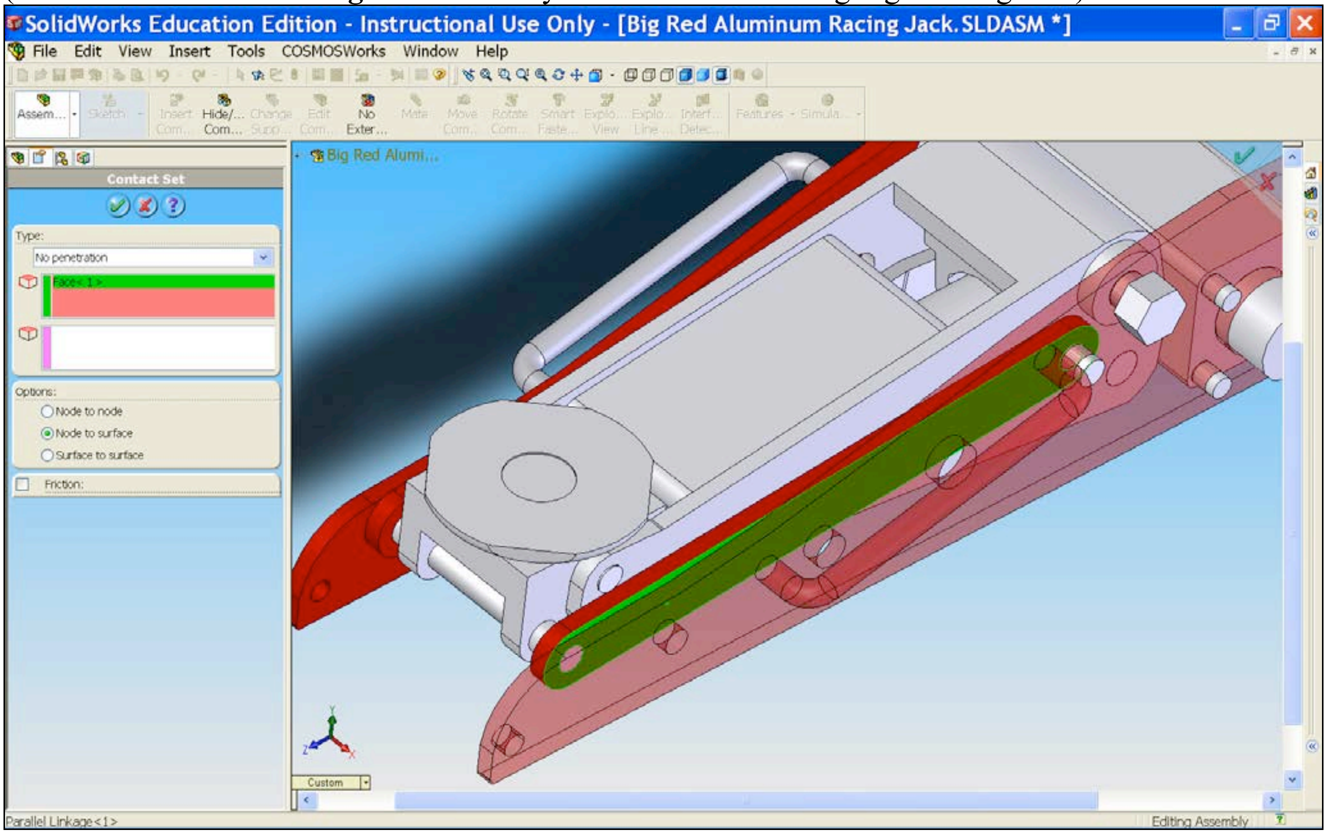
69) Now, let's fix the interference problem of the parts with the Left Frame. Put the mouse pointer anywhere on the left frame surface and right-click it. Then, select  **Change Transparency**.



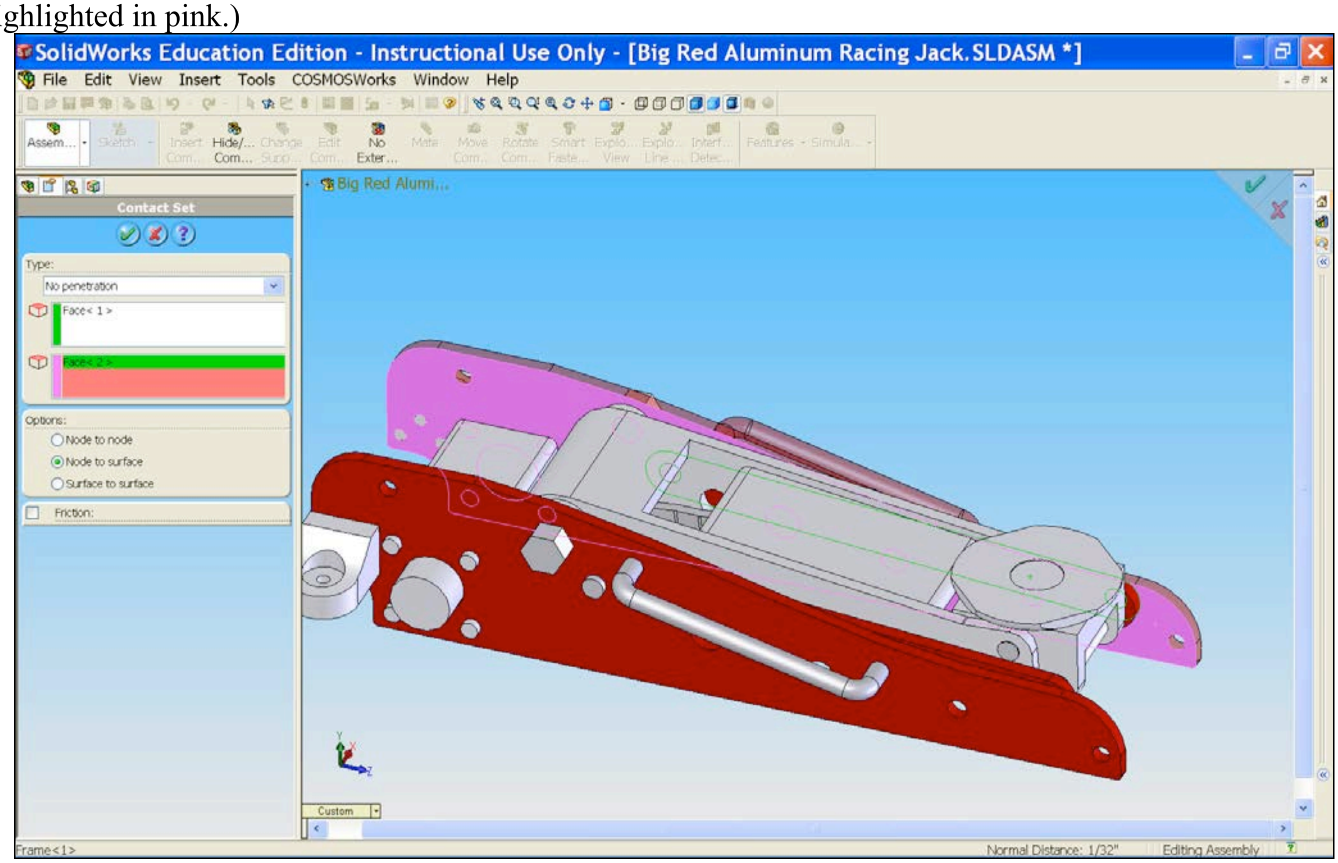
70) The left frame is now transparent Next, right-click  **Contact/Gaps** and select **Define Contact Set**.



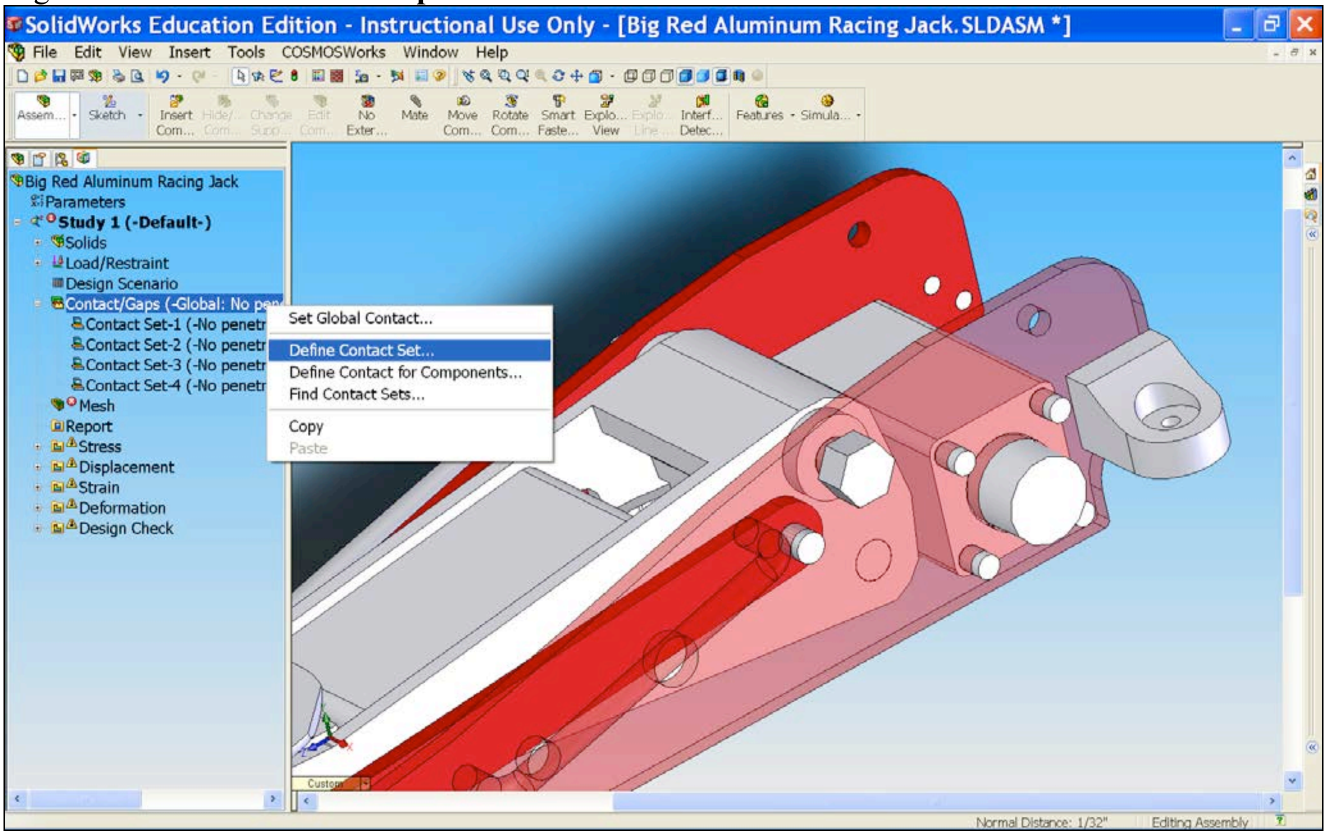
71) For  **Faces, Edges, Vertices for Source,** select the face of **Parallel Linkage** as shown in picture below. (Note: The **Parallel Linkage** surface that you need to select is highlighted in green.)



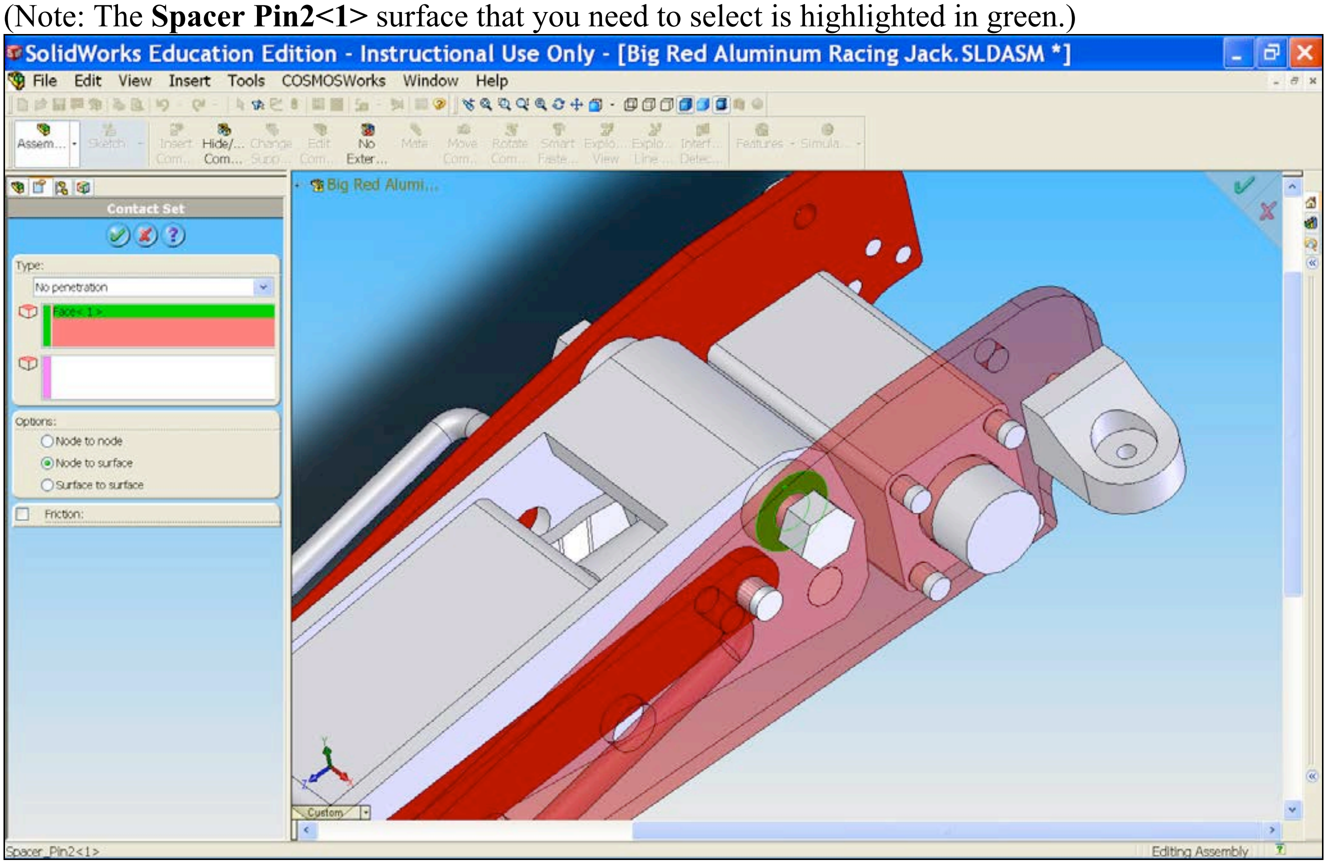
72) For  **Faces for Target**, select the surface of the **Left Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Left Frame** surface that you need to select is highlighted in pink.)



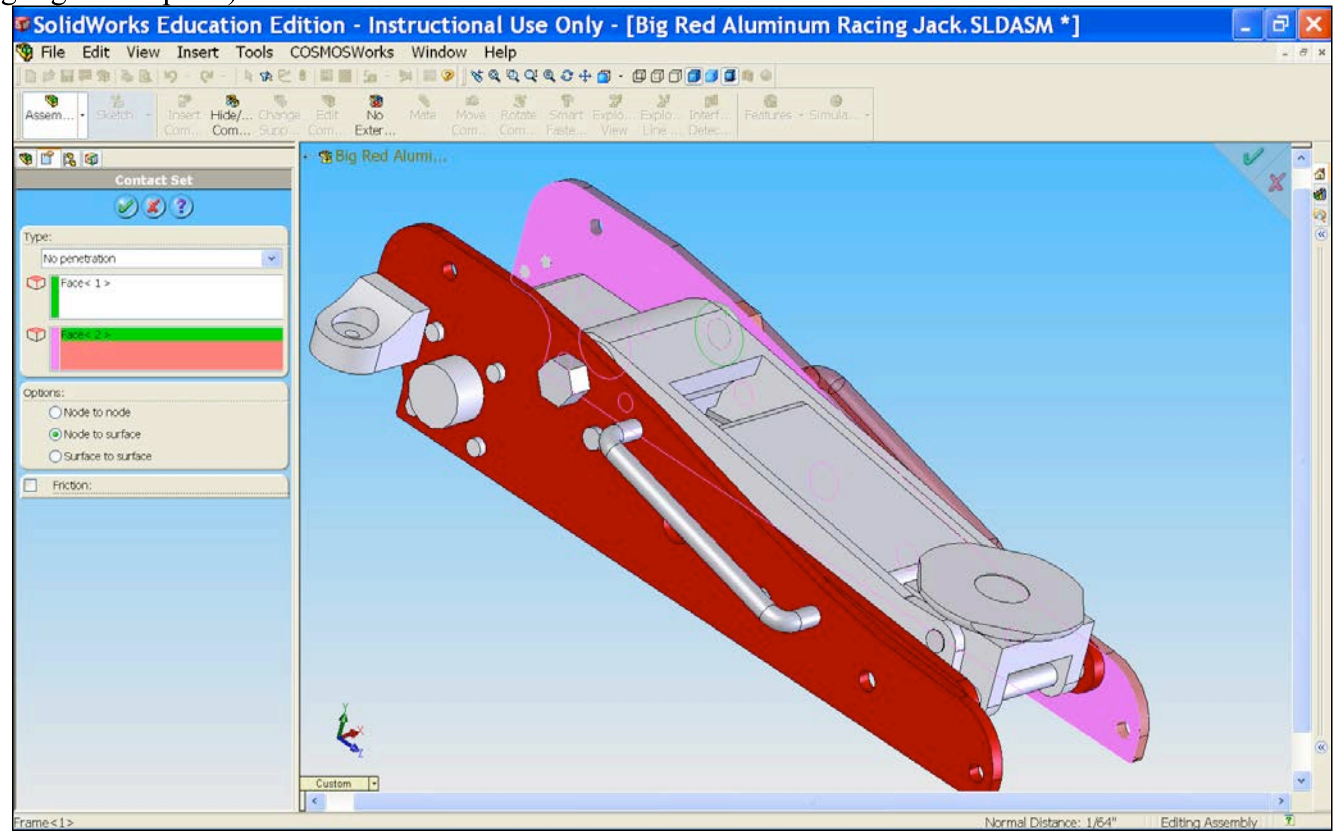
73) Right-click  **Contact/Gaps** and select **Define Contact Set**.



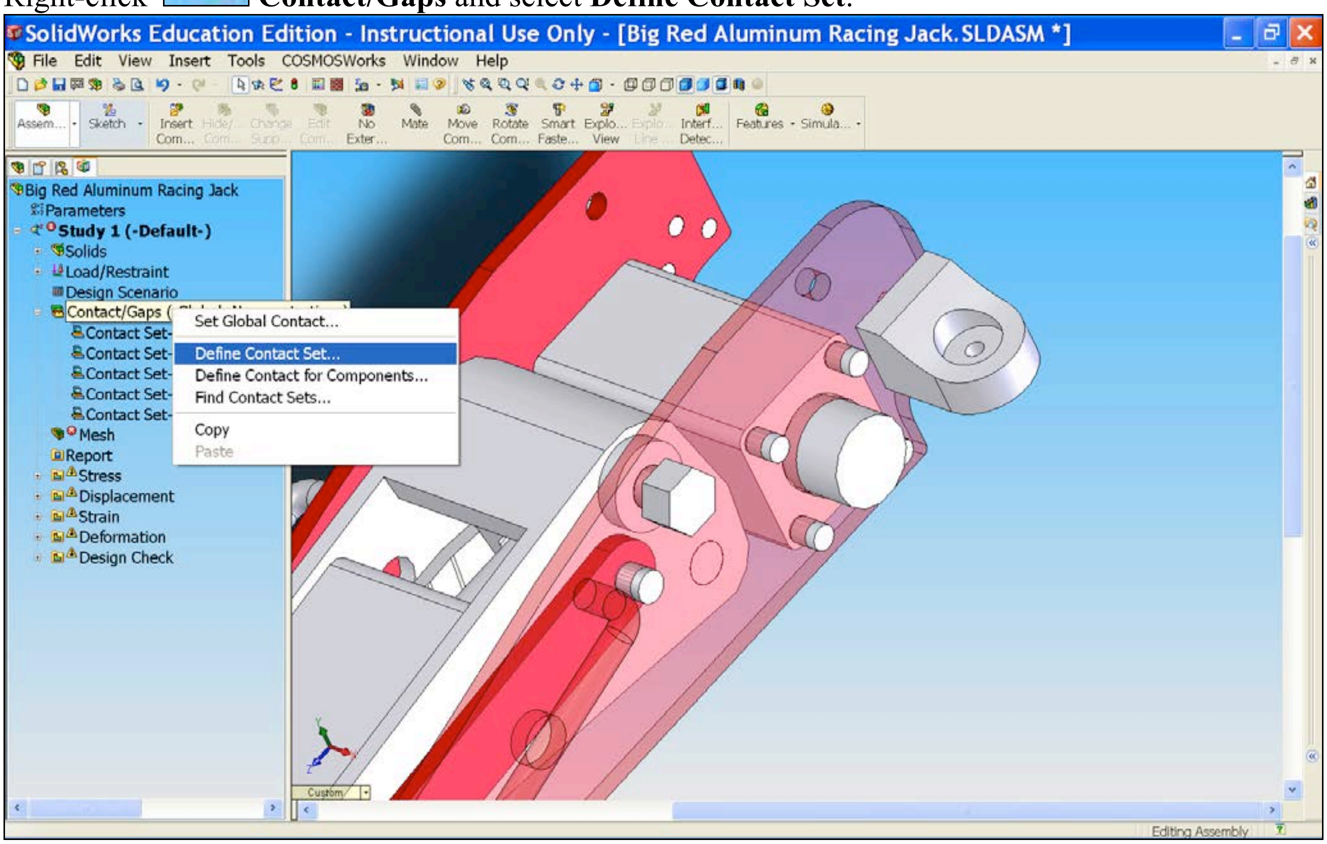
74) For  **Faces, Edges, Vertices for Source,** select the face of **Spacer Pin2<1>** as shown in picture below. (Note: The **Spacer Pin2<1>** surface that you need to select is highlighted in green.)

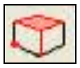


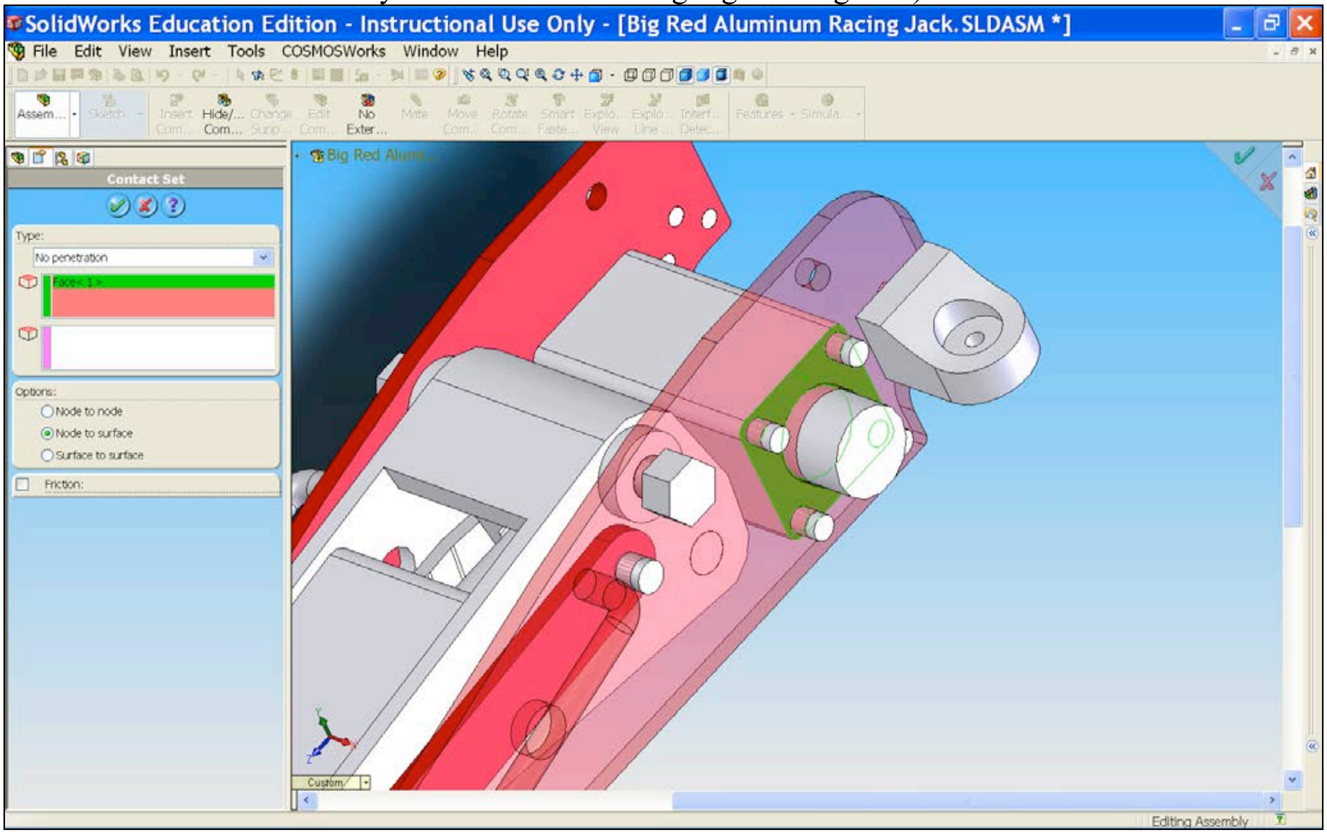
75) For  **Faces for Target**, select the surface of the **Left Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Left Frame** surface that you need to select is highlighted in pink.)



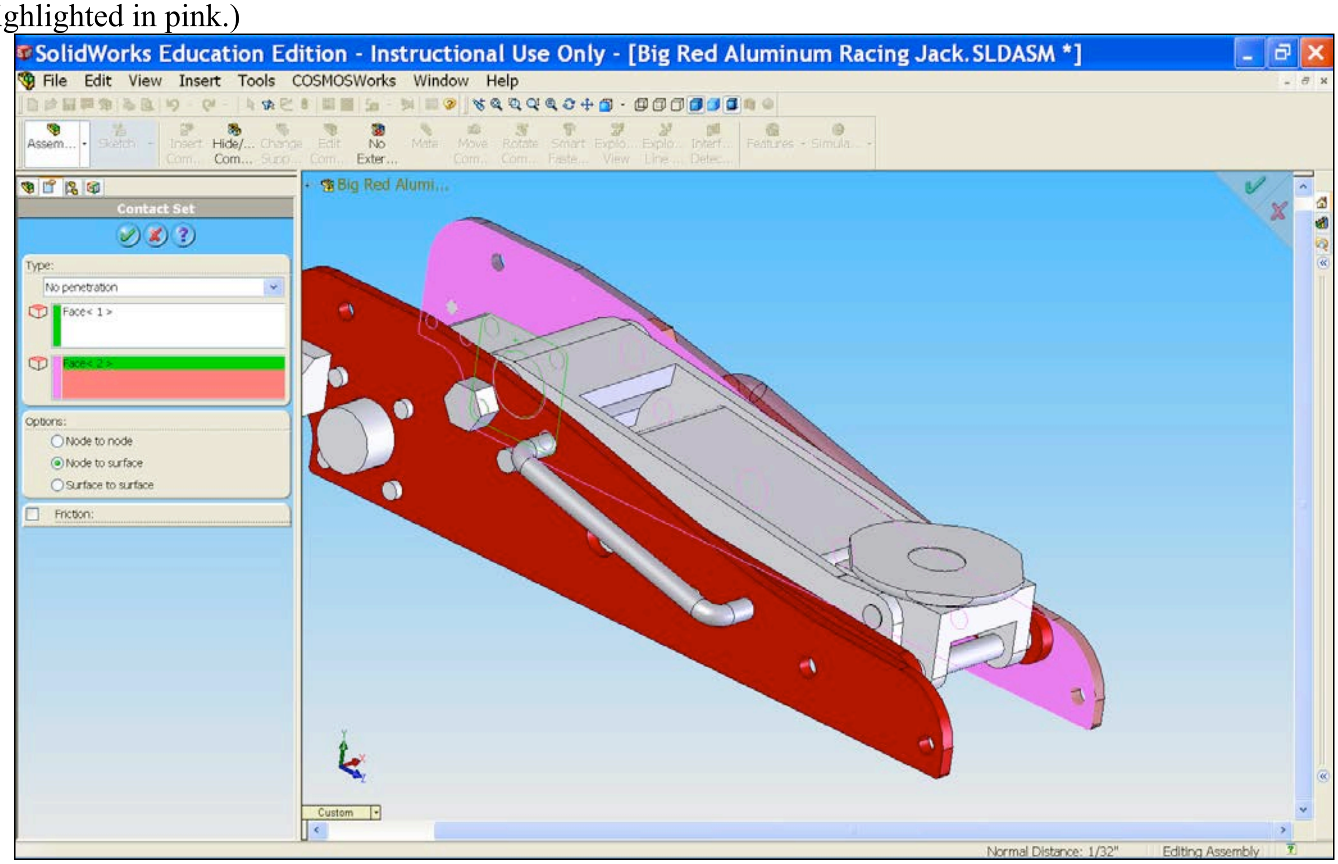
76) Right-click  **Contact/Gaps** and select **Define Contact Set**.



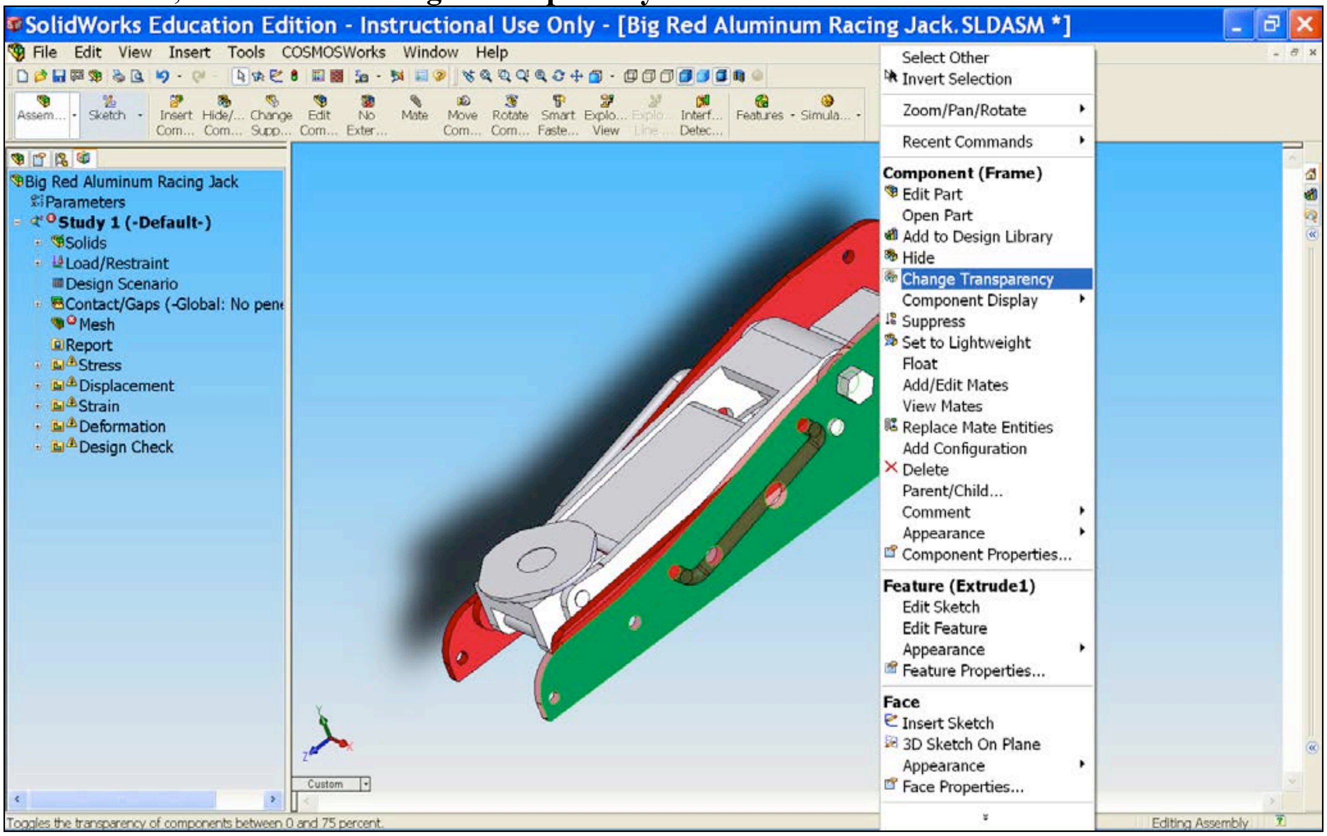
77) For  **Faces, Edges, Vertices for Source,** select the face of **Bottle Jack** as shown in picture below. (Note: The **Bottle Jack** surface that you need to select is highlighted in green.)



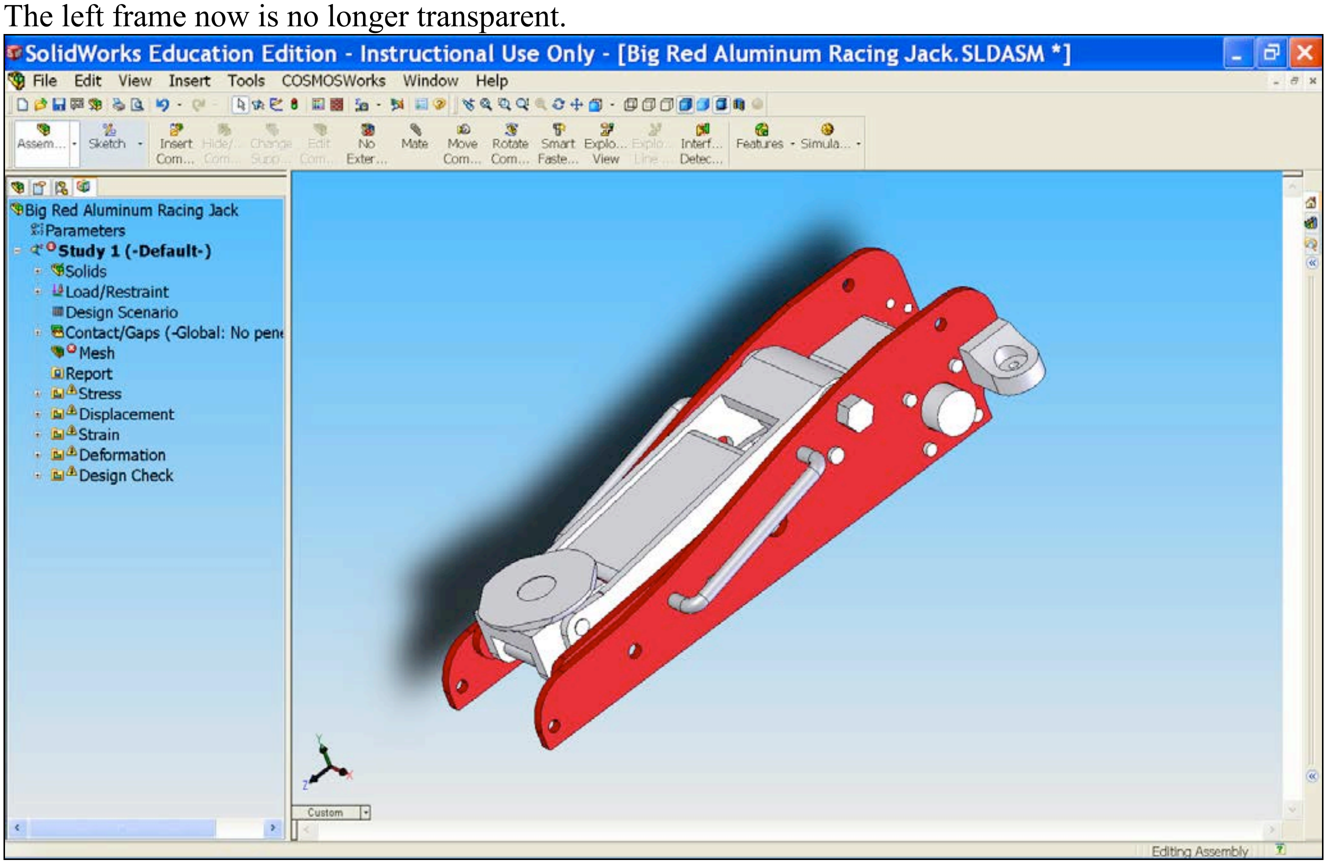
78) For  **Faces for Target**, select the surface of the **Left Frame** as shown in the picture below. Under **Options**, select **Node to Surface**. Click **OK**. (Note: The **Left Frame** surface that you need to select is highlighted in pink.)



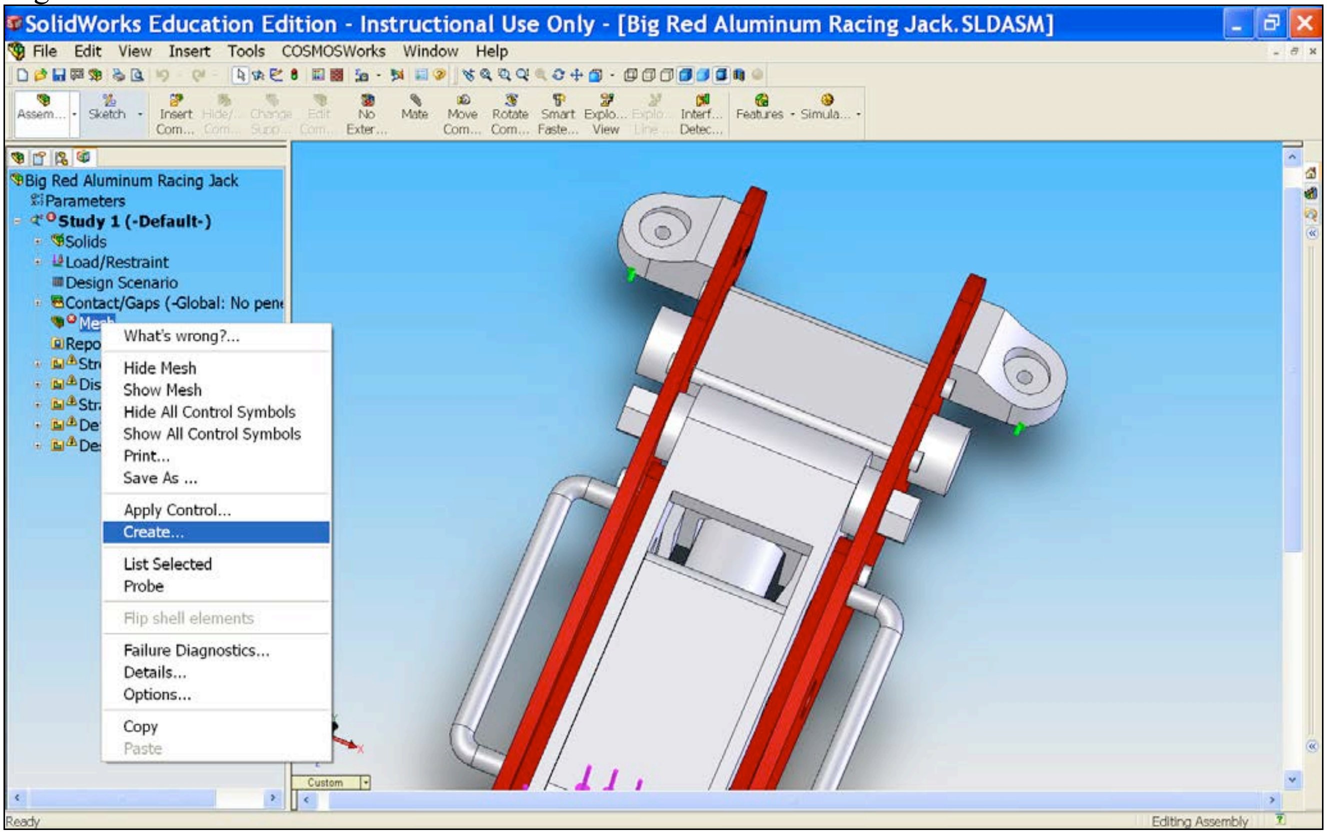
79) We have completed defining gaps to avoid interference of the parts with the left frame. We can now change the transparency of the left frame back to normal. Put the mouse pointer anywhere on left frame surface and right-click it. Then, select  **Change Transparency**.



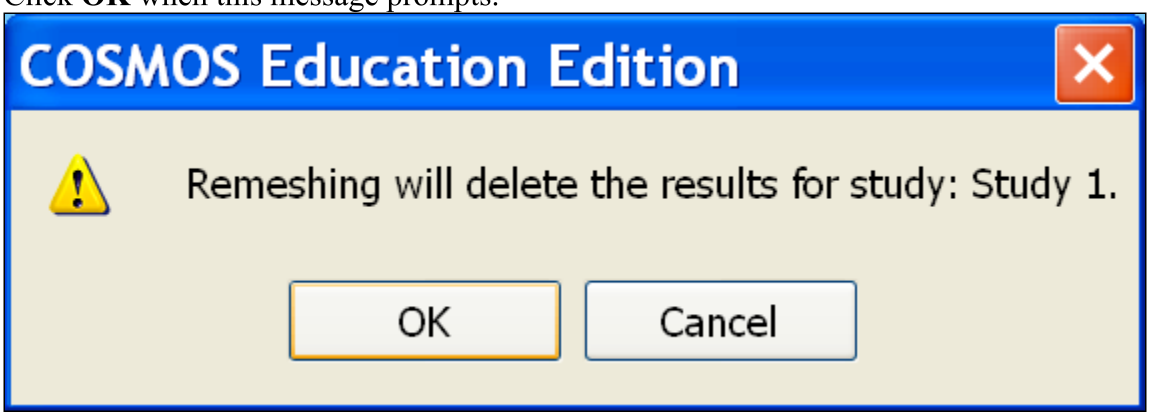
80) The left frame now is no longer transparent.



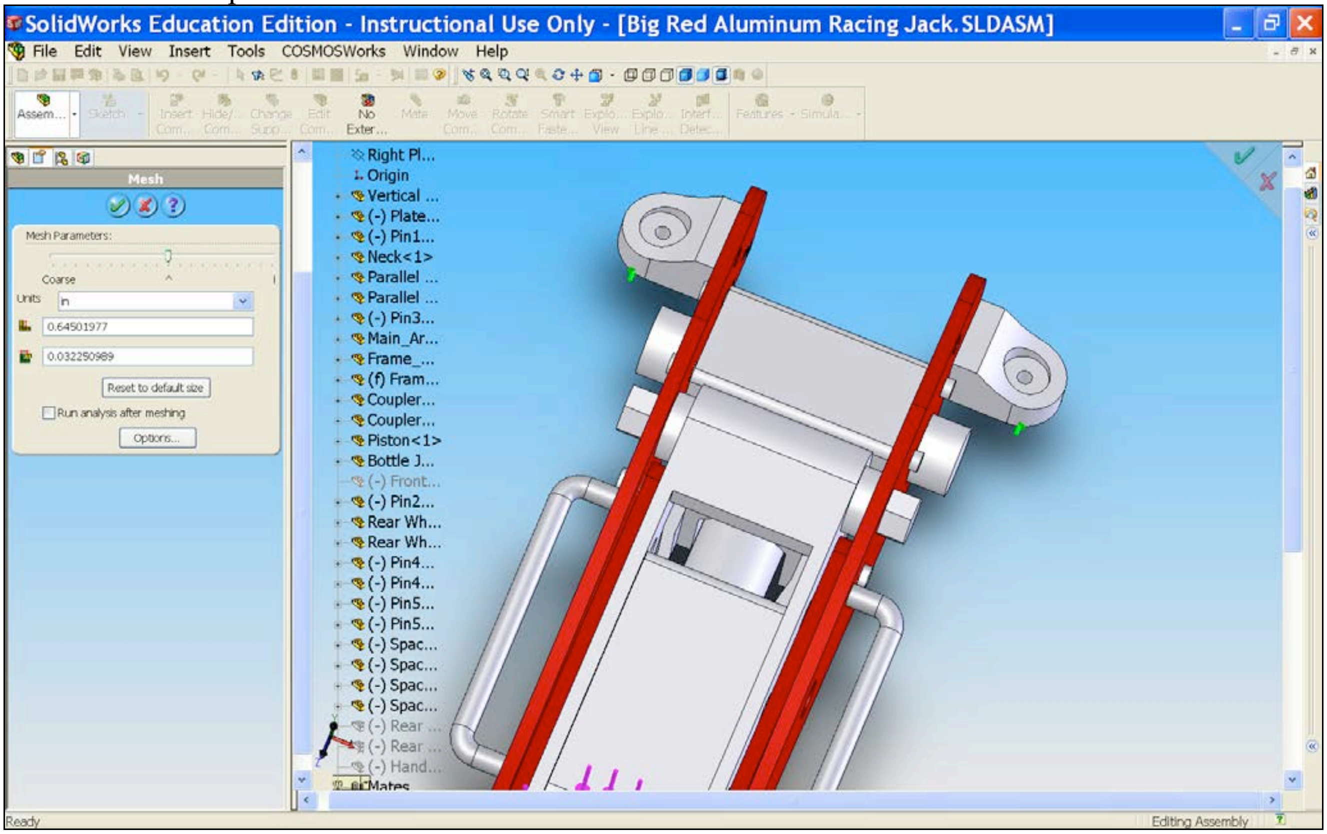
81) Right-click **Mesh** and select **Create**.



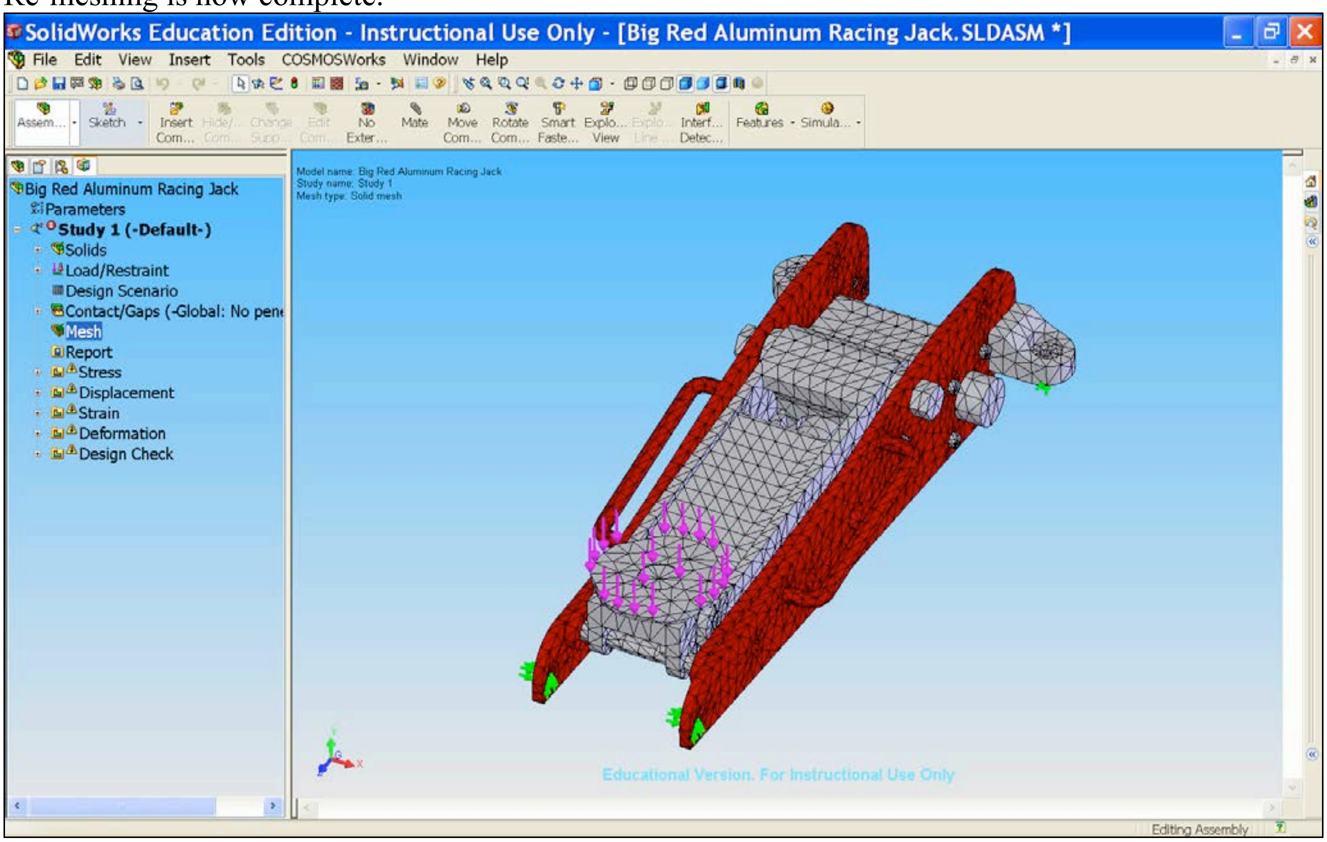
82) Click **OK** when this message prompts.



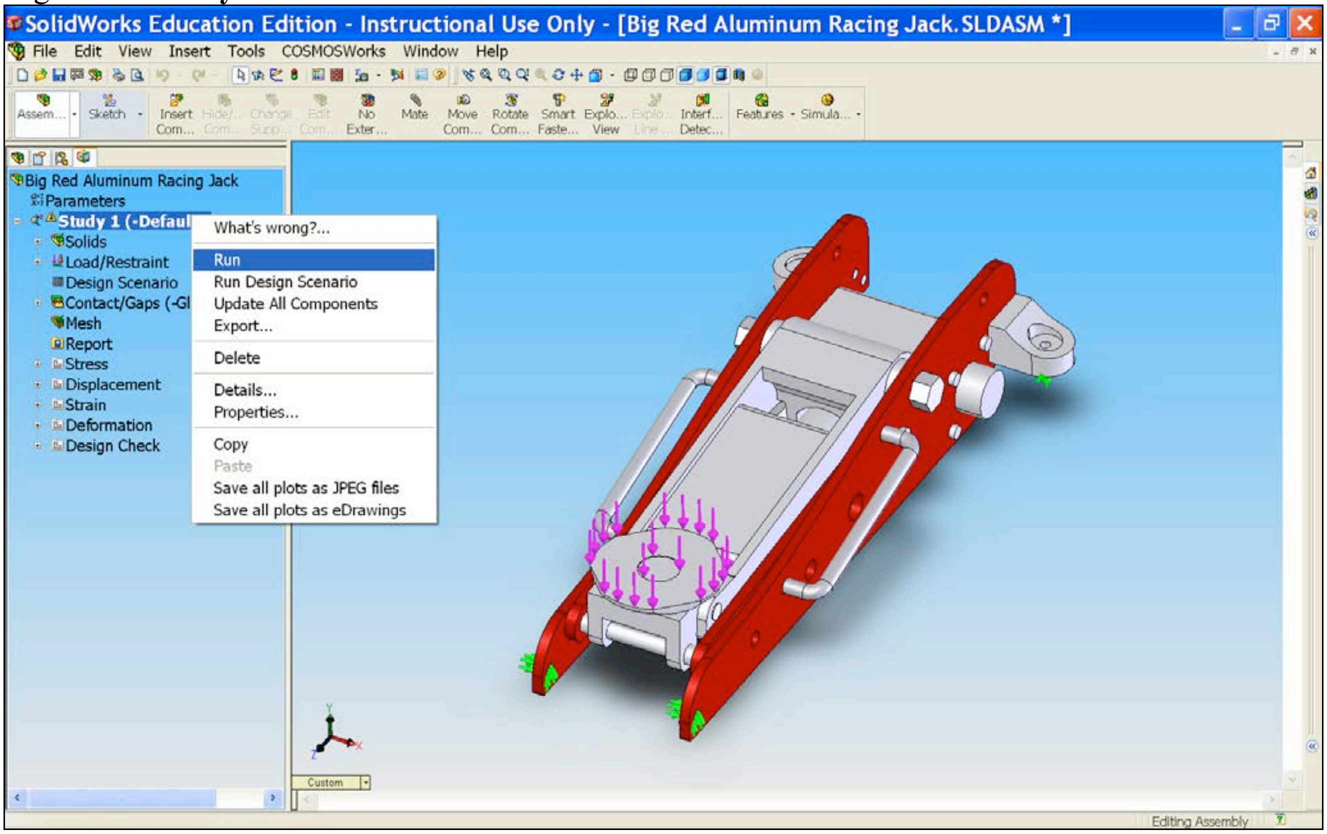
83) Click **OK** to accept the default values.



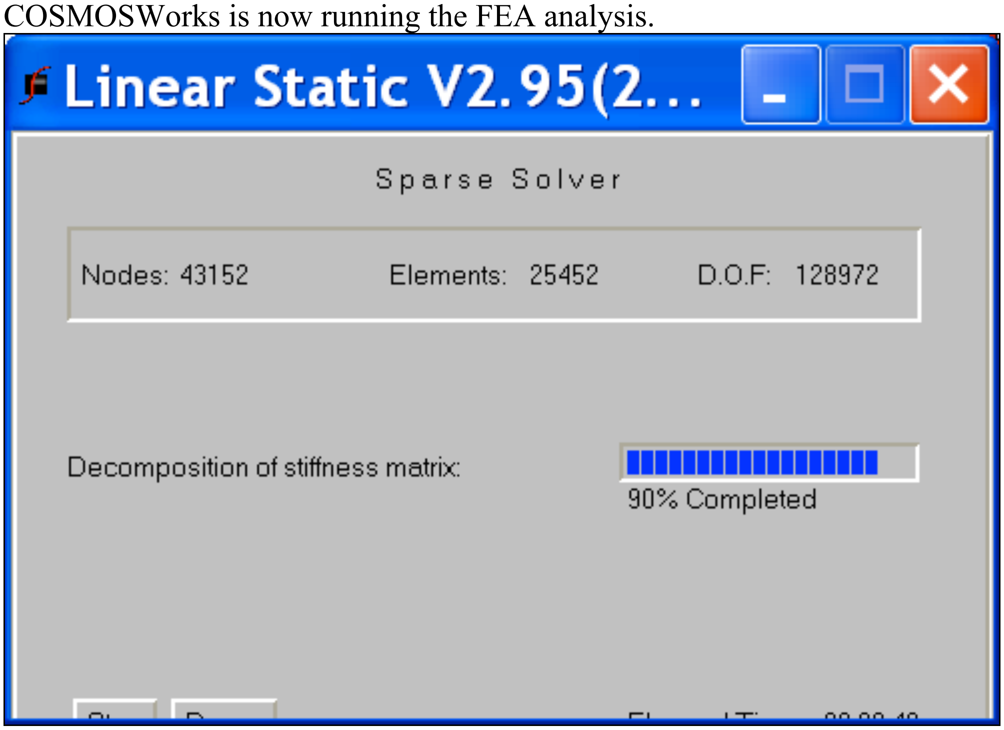
84) Re-meshing is now complete.



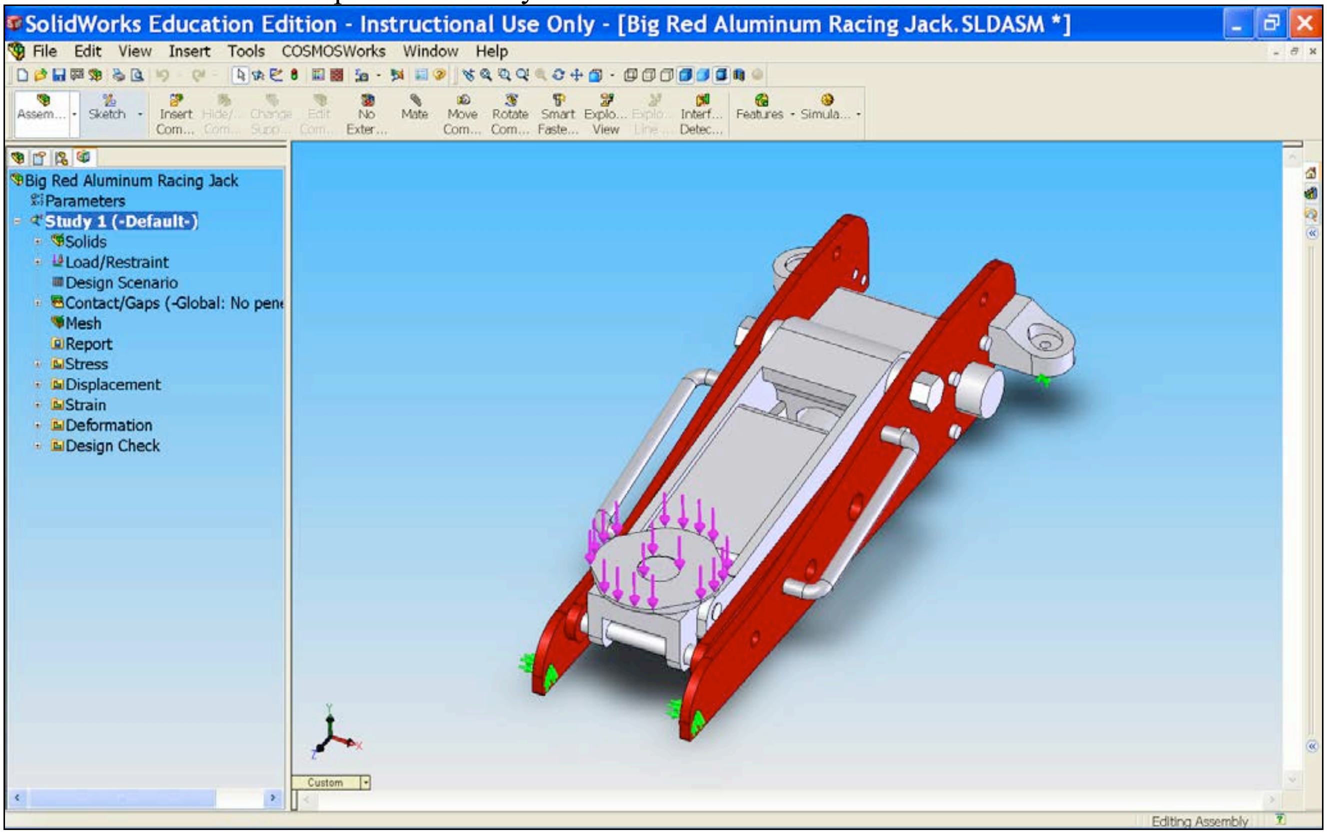
85) Right-click **Study 1** and select **Run**.



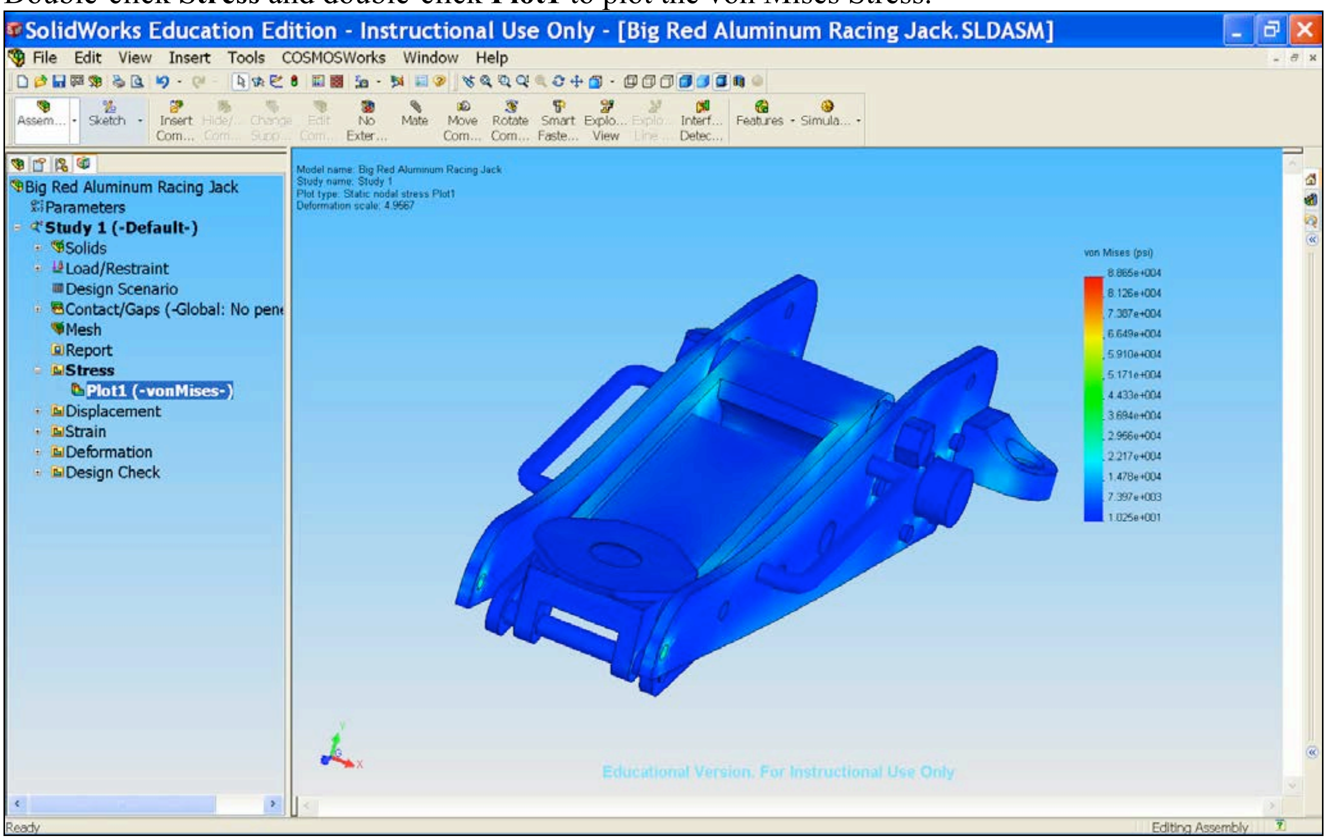
86) COSMOSWorks is now running the FEA analysis.



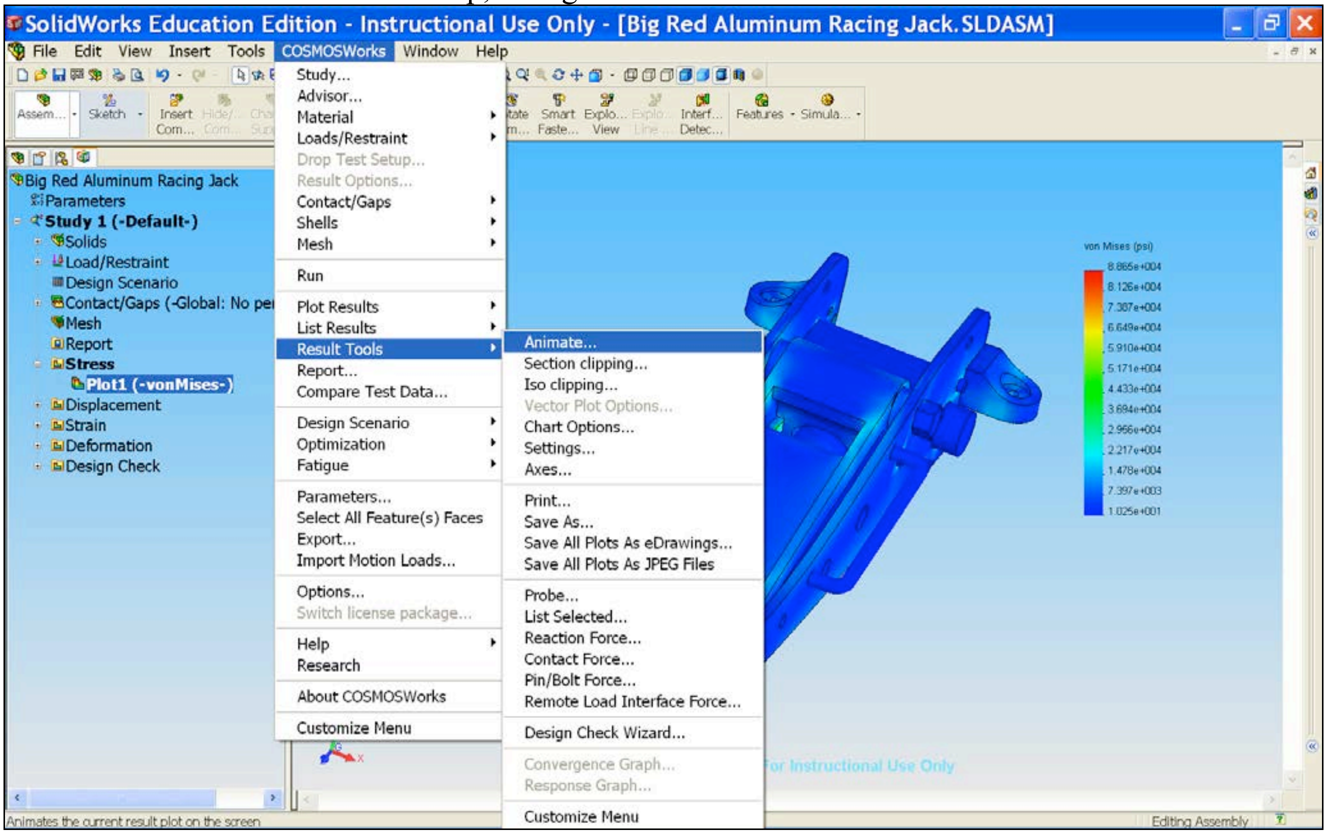
87) COSMOSWorks has completed the analysis.



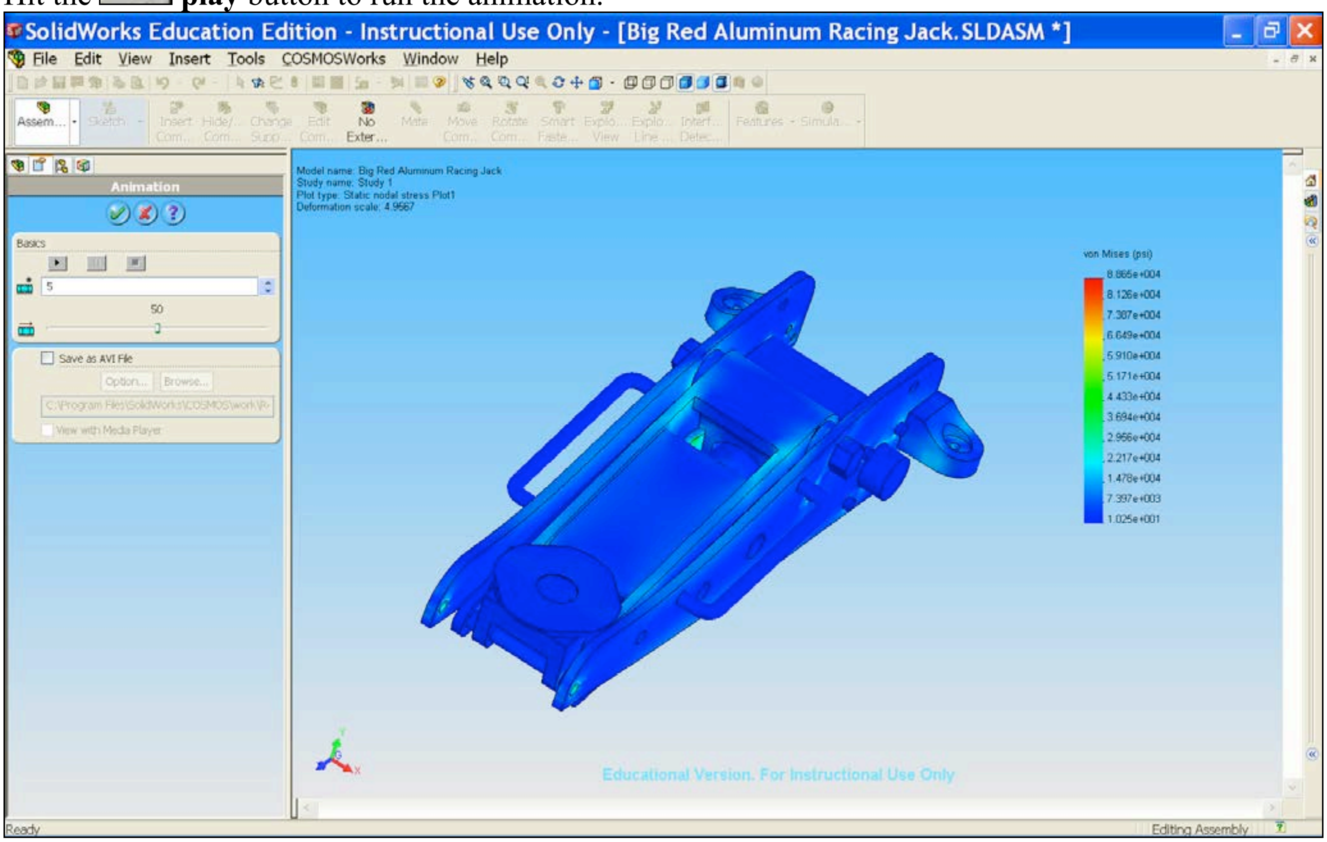
88) Double-click **Stress** and double-click **Plot1** to plot the von Mises Stress.



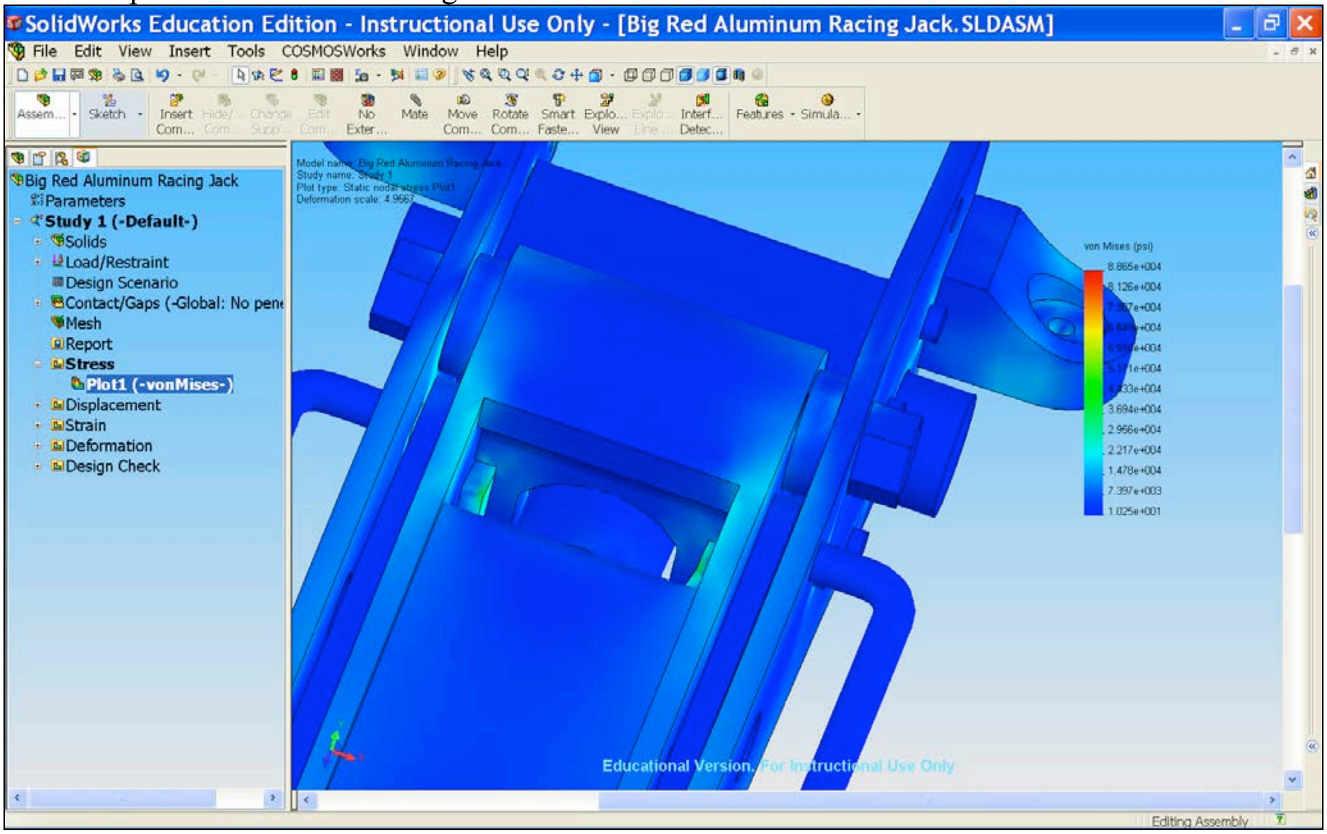
89) Go to **COSMOSWorks** menu on top, then go to **Result Tools** and select **Animate**.



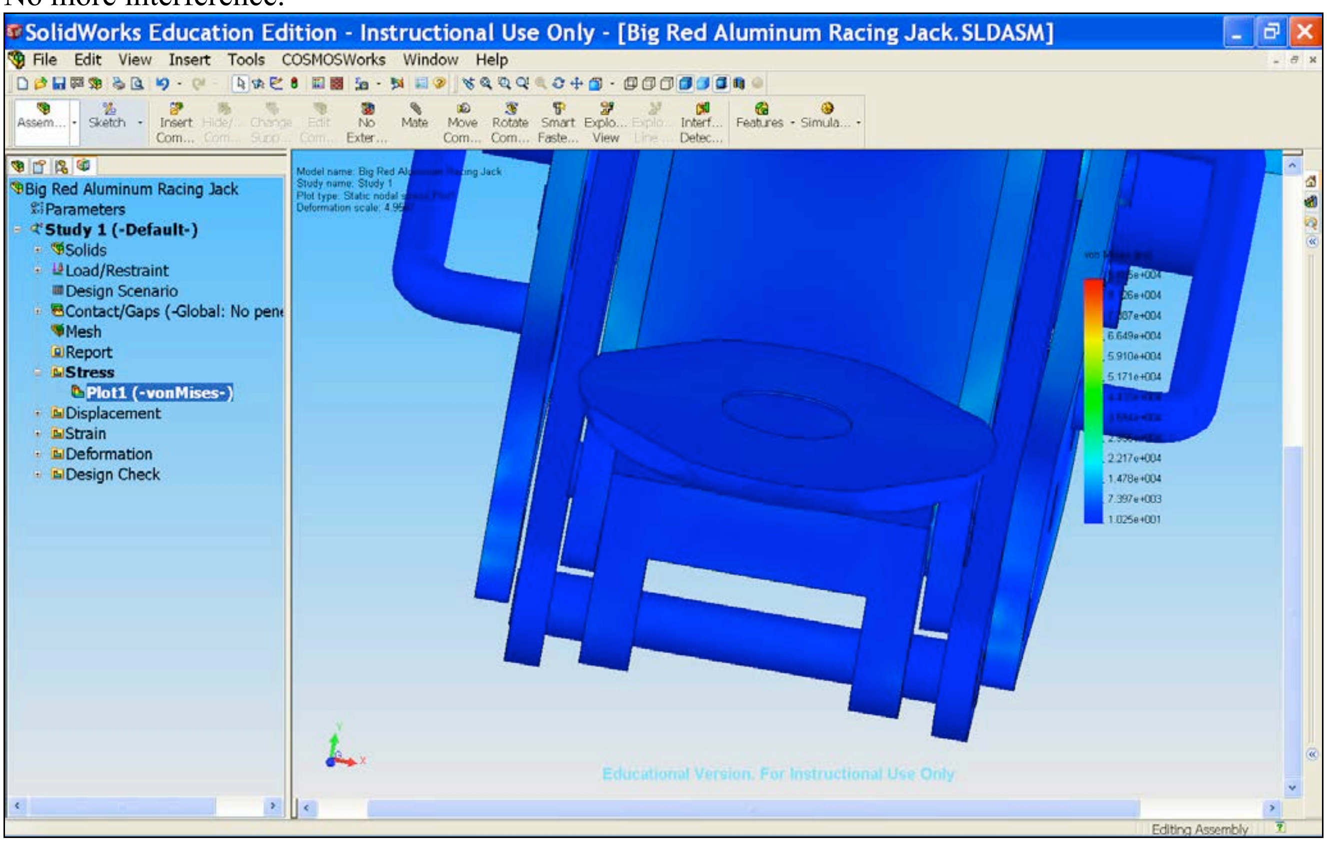
90) Hit the  **play** button to run the animation.



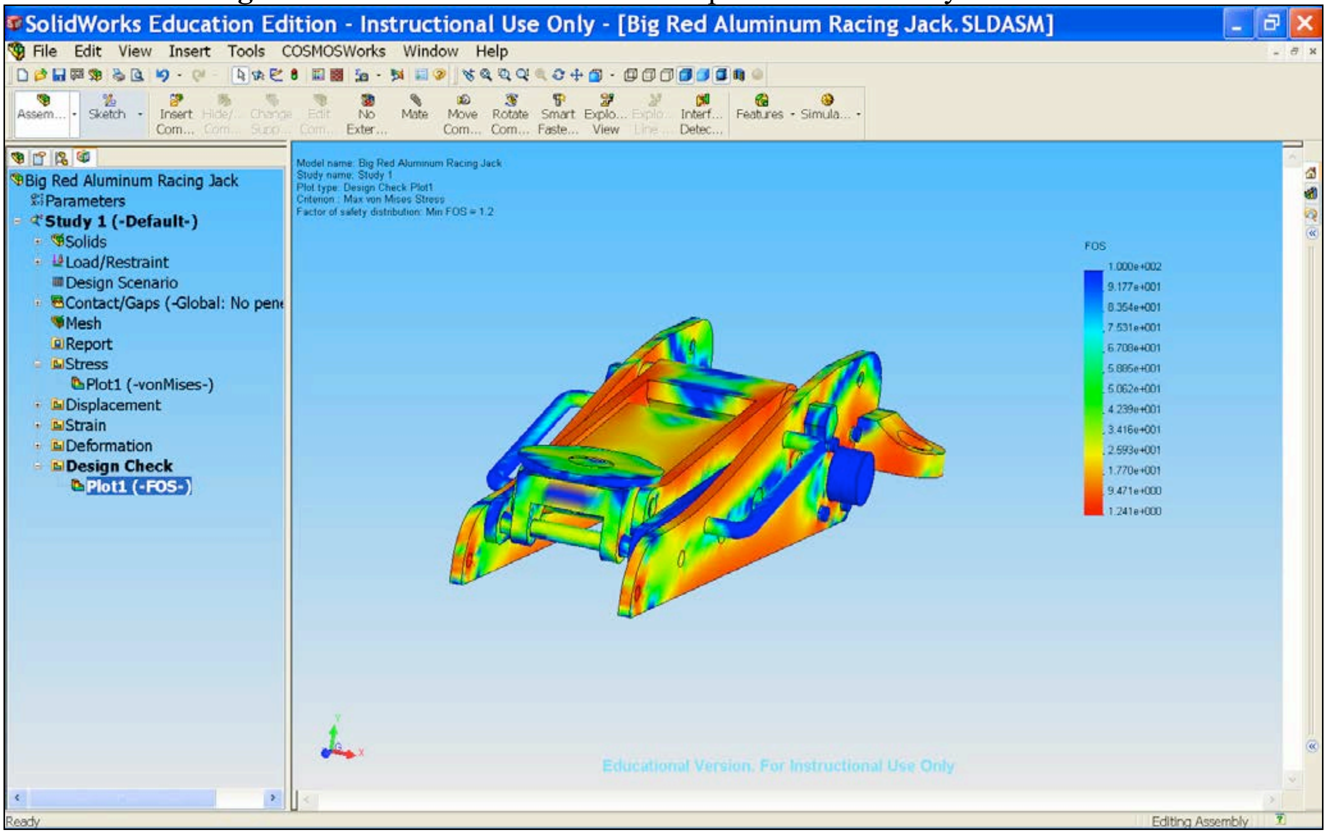
91) Note that parts interference no longer occurs.



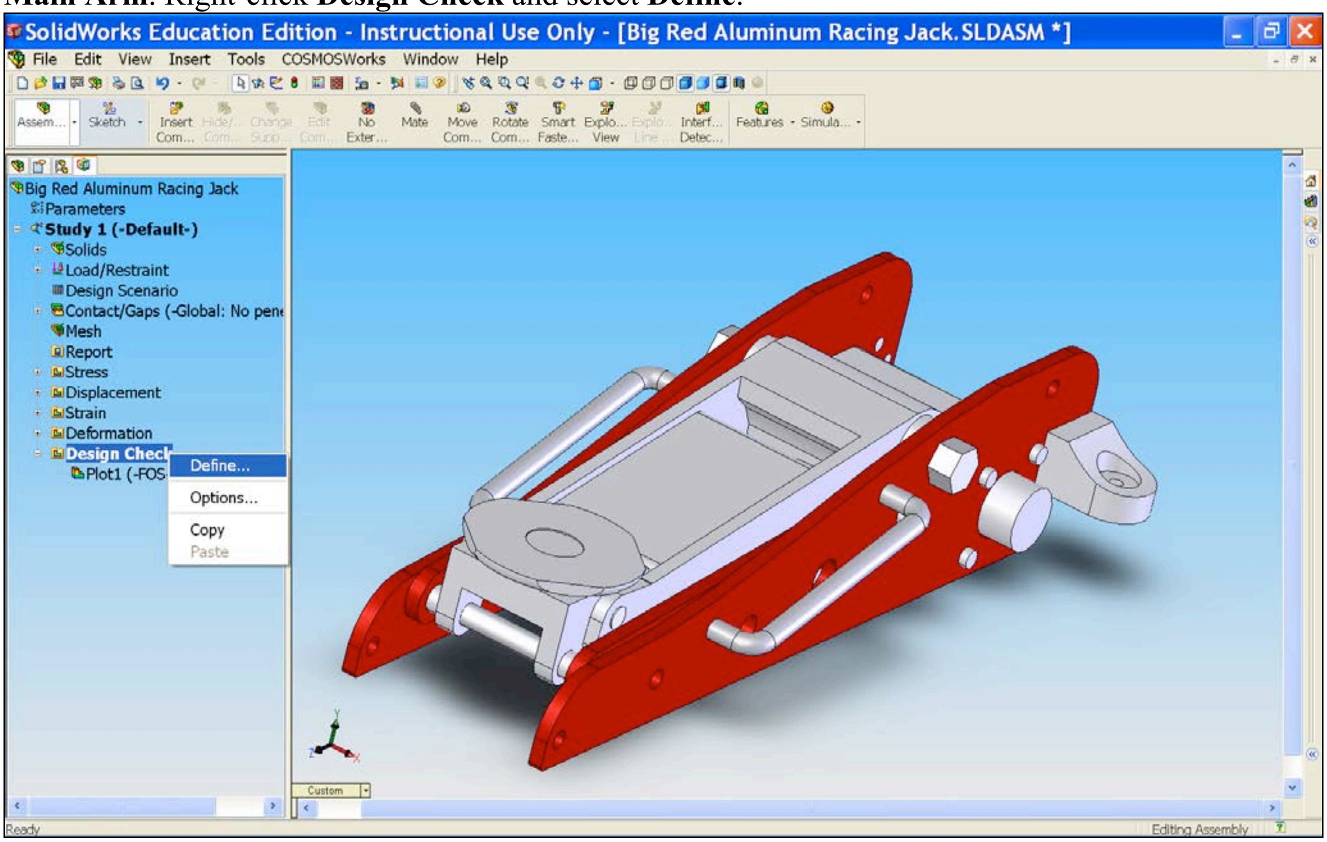
92) No more interference.



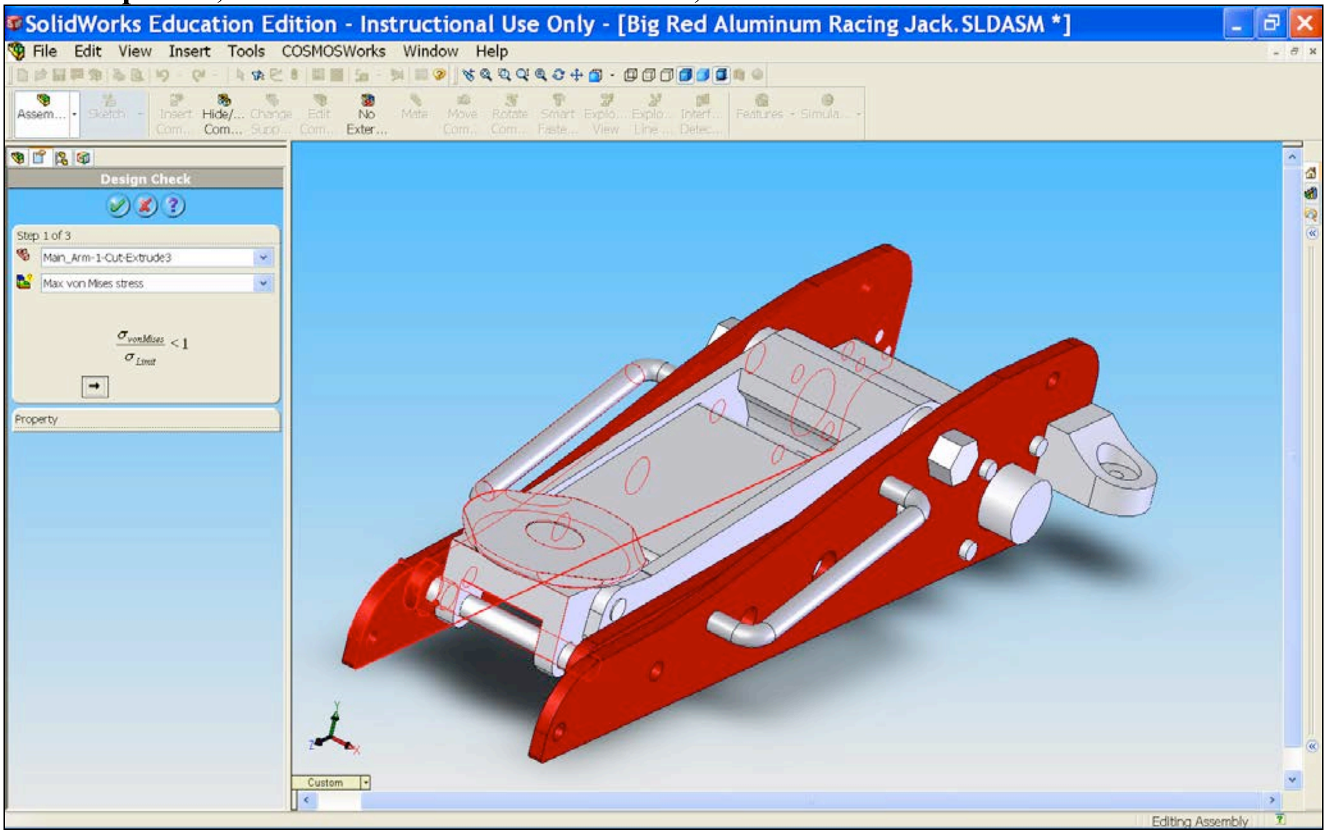
93) Double-click **Design Check** and double-click **Plot1** to plot factor of safety.



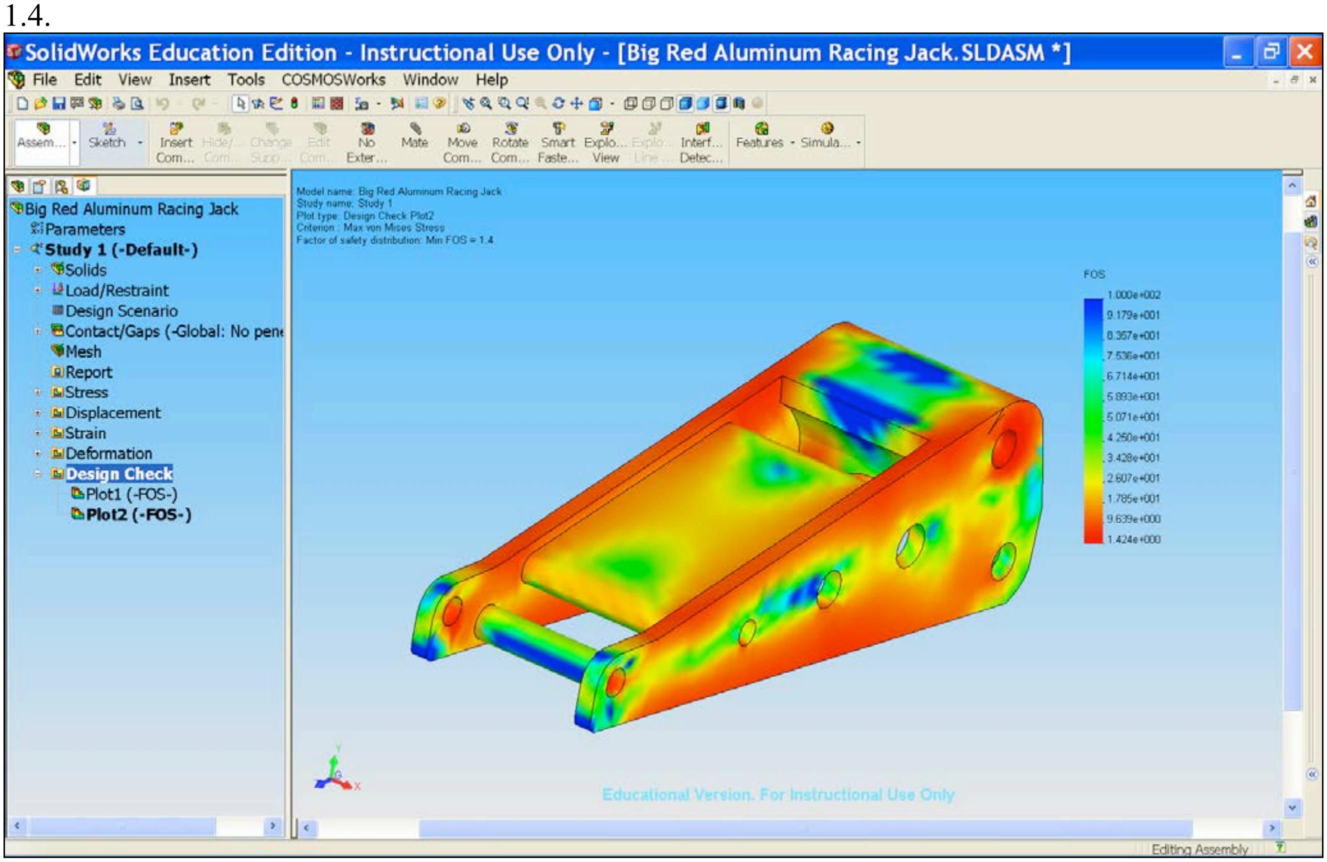
94) Now, let us find out the safety factor of individual component. Since the **Main Arm** is one of the most important parts that determine how much load the floor jack can support, let us plot the safety factor of the **Main Arm**. Right-click **Design Check** and select **Define**.



95) For **Component**, select **Main Arm** and for **Criterion**, select **Max von Mises Stress**. Click **OK**.



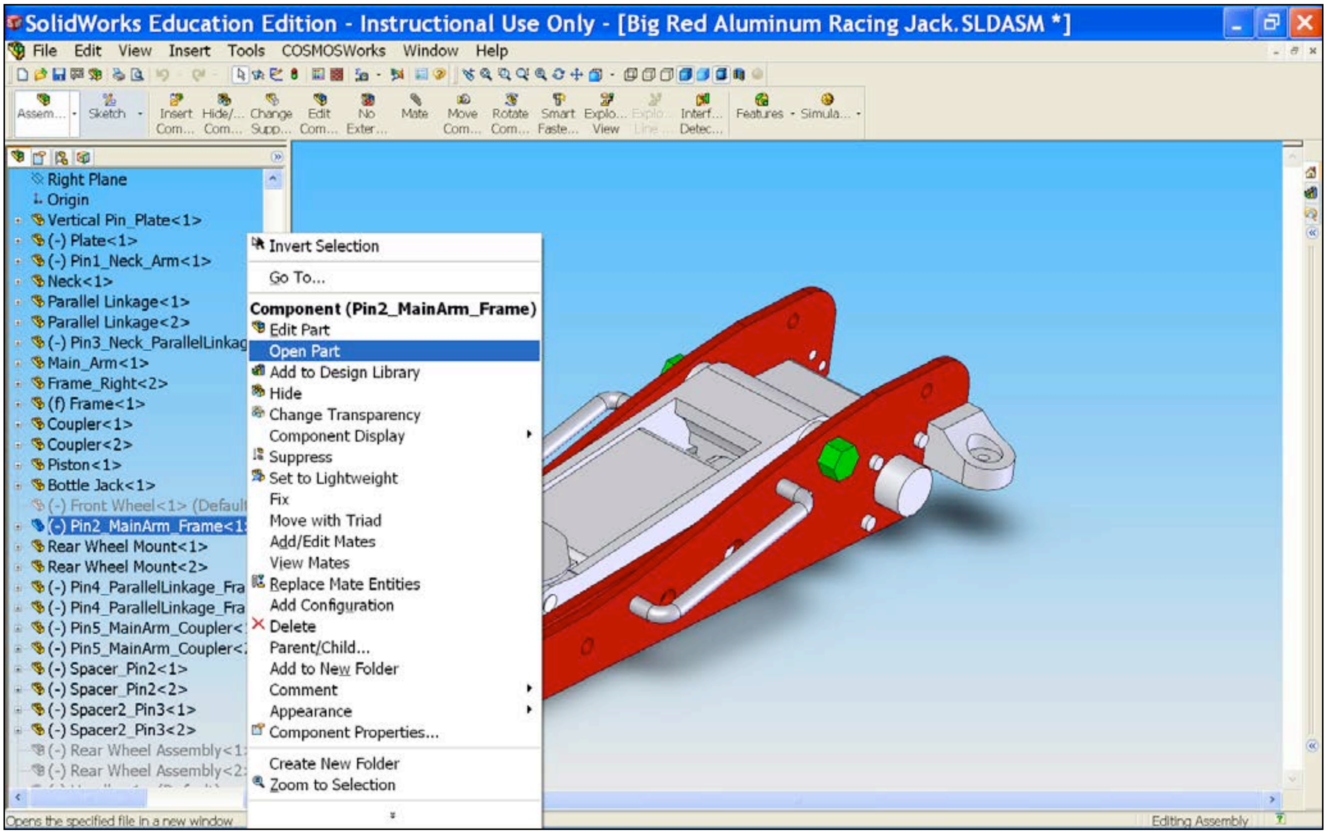
96) The safety factor of the **Main Arm** is plotted and the result shows that the factor of safety of the **Main Arm** is 1.4.



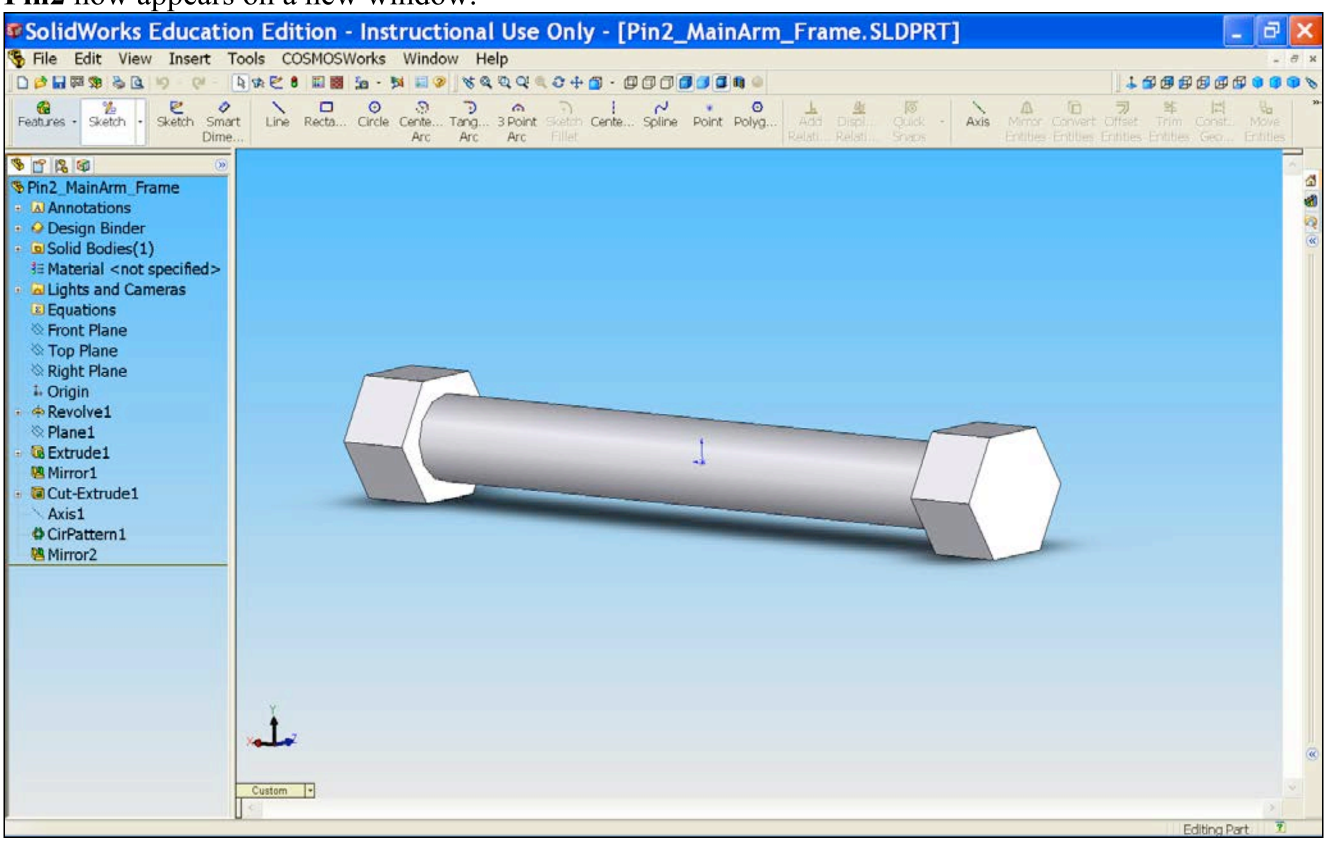
97) This floor jack assembly was modeled with a parameterized function that allows the user to modify the geometry of certain component and having the SolidWorks to automatically change the geometry of the other components associated with the component which its geometry was manually modified. For instance, if you

change the diameter of a pin, the hole in which the pin is inserted into is automatically modified with respect to the change in the pin diameter.

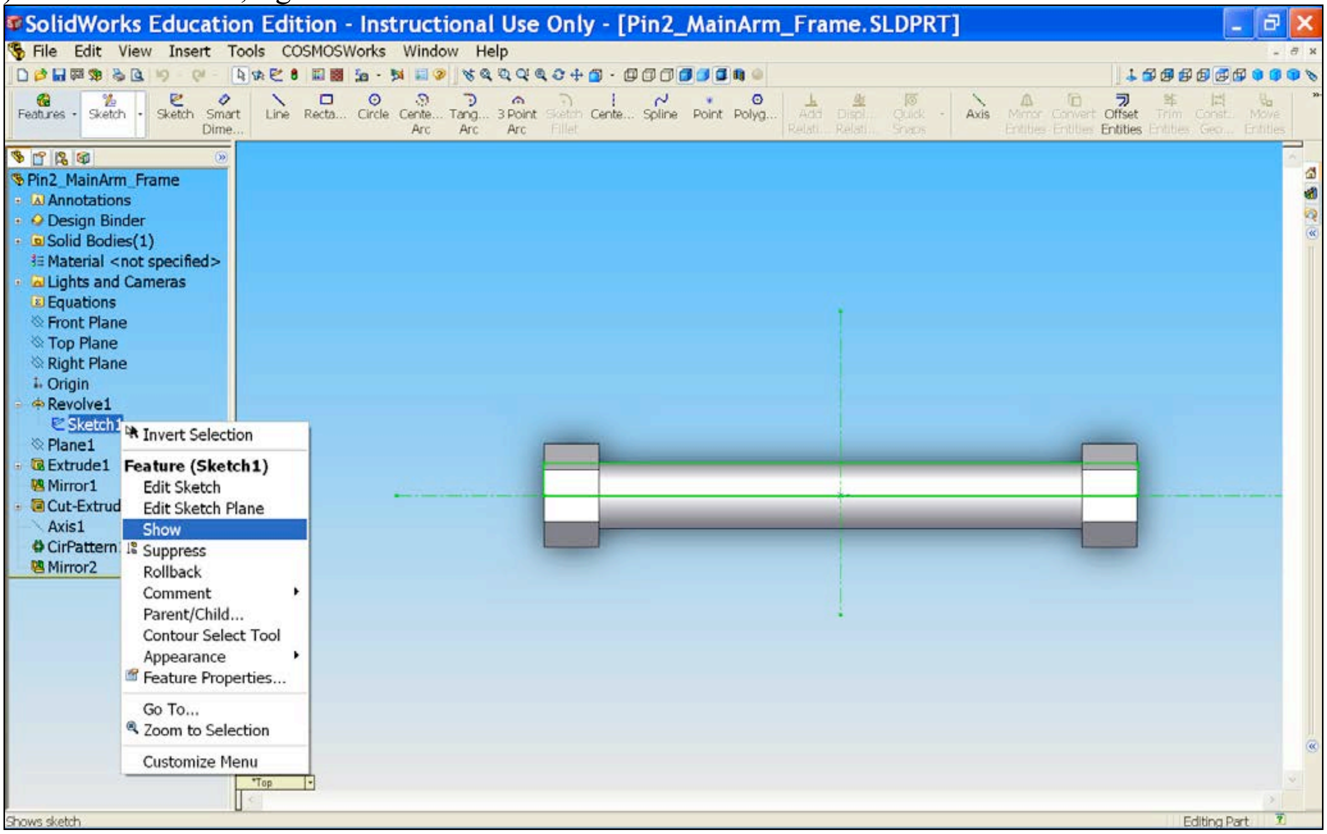
98) Now, let us change a radius of **Pin2** to utilize the parameterized function. Right-click **Pin2** and select **Open Part**.



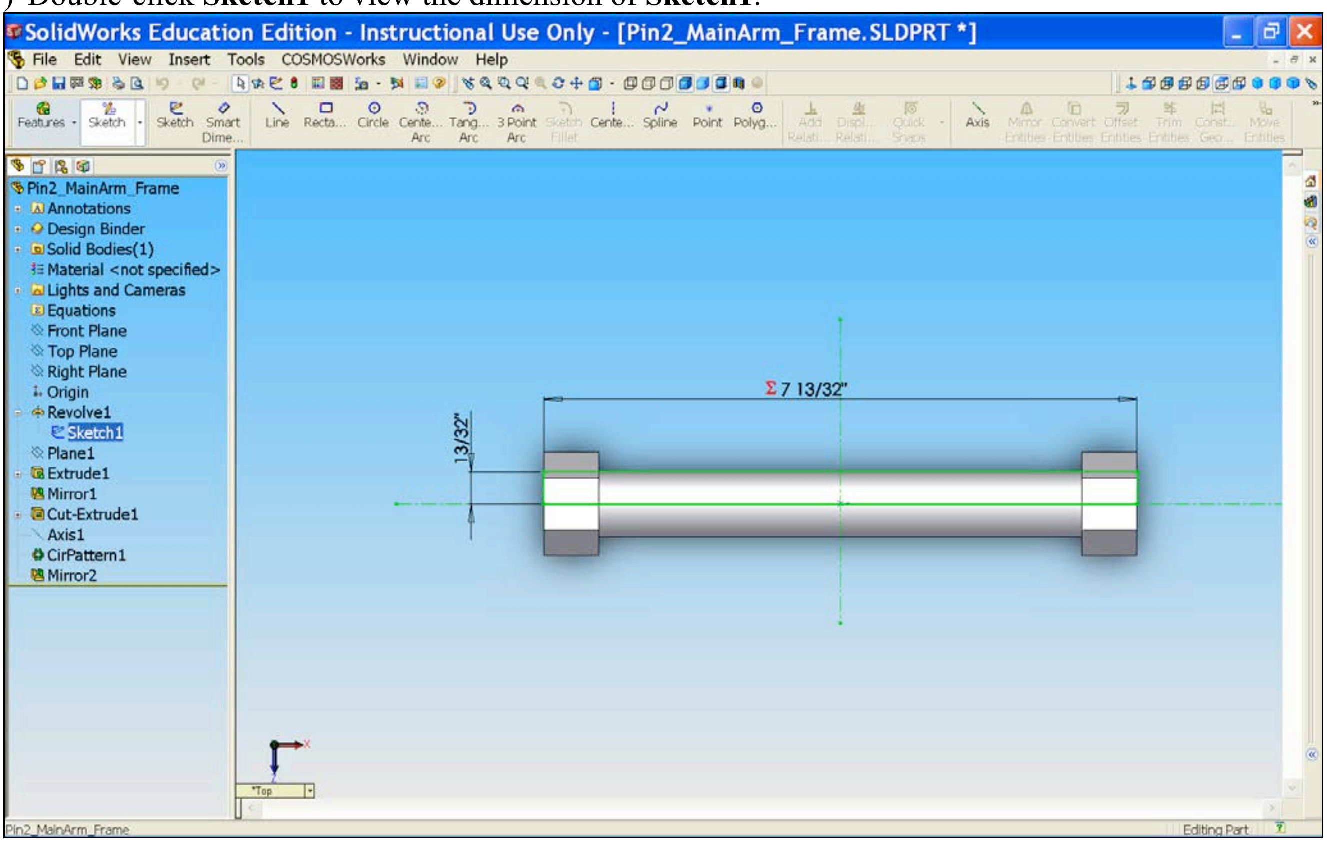
99) **Pin2** now appears on a new window.



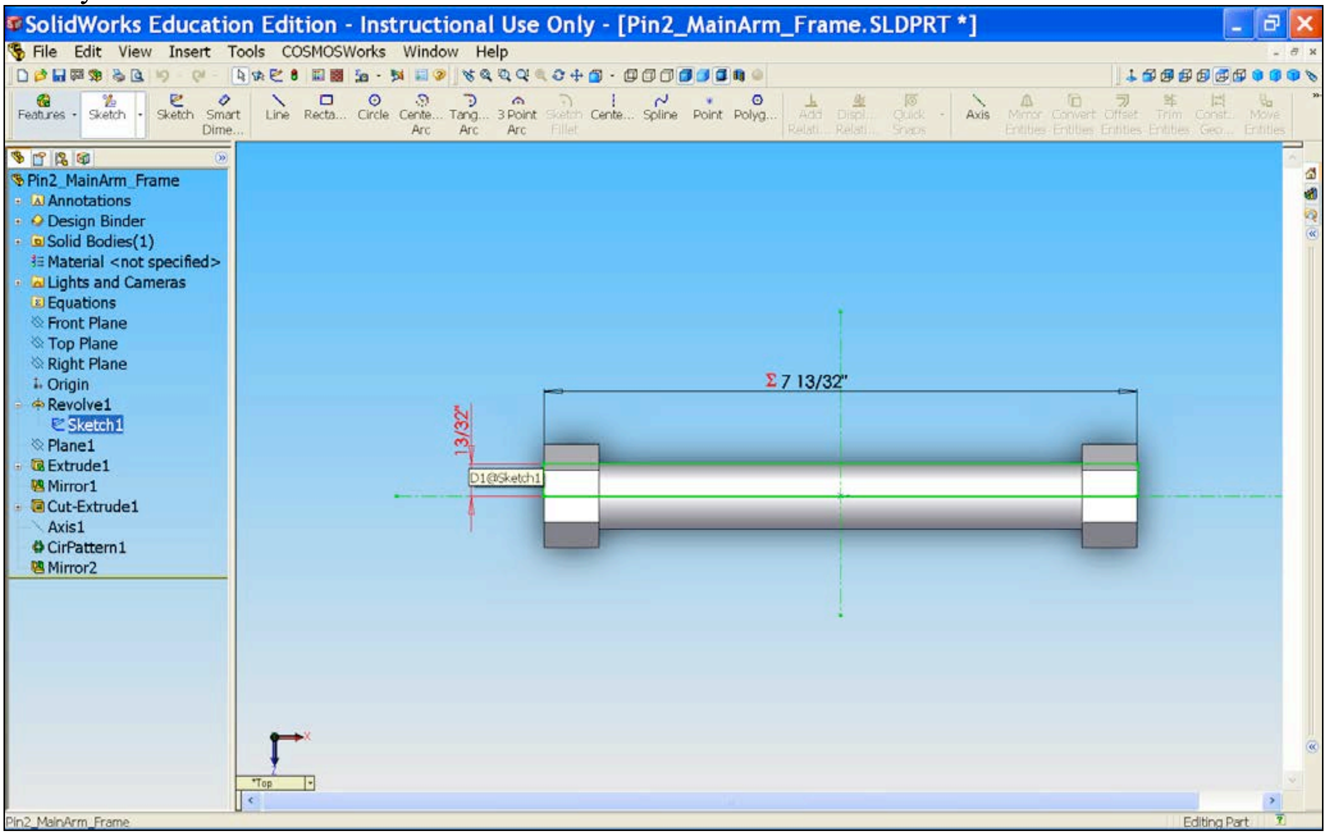
100) Under **Revolve1**, right-click **Sketch1** and select **Show**.



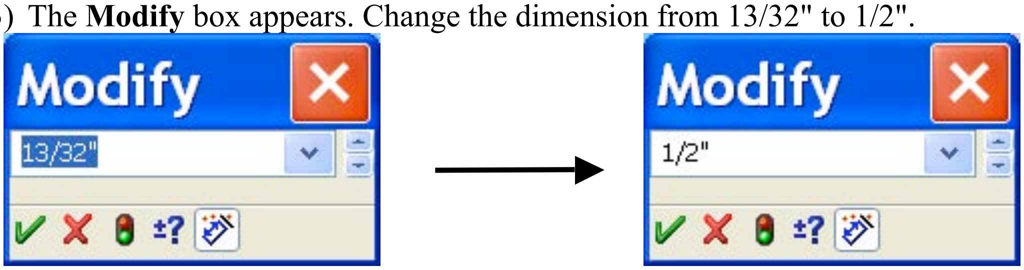
101) Double-click **Sketch1** to view the dimension of **Sketch1**.

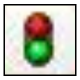


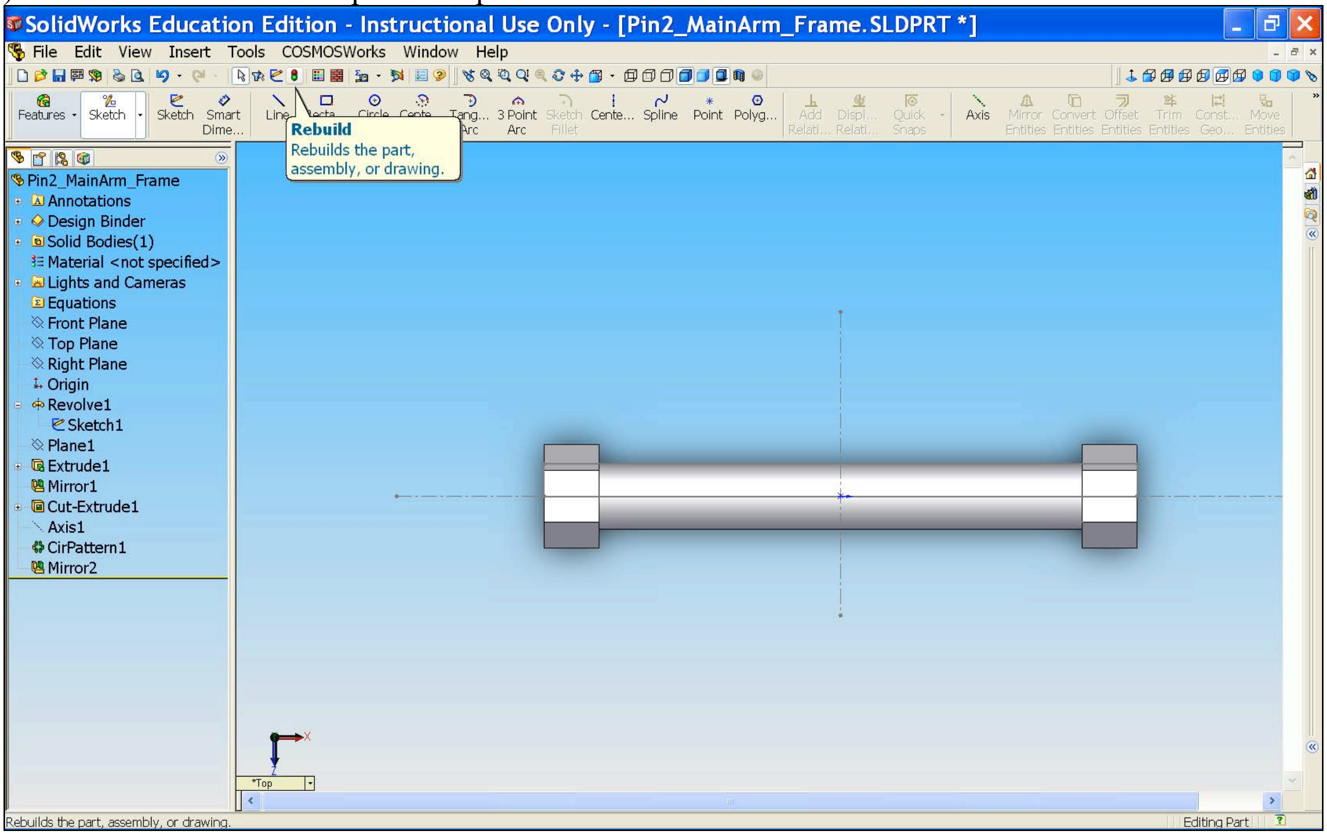
102) Place the mouse pointer on the dimension noted as **D1@Sketch1**. Then, double-click that dimension to modify its value.



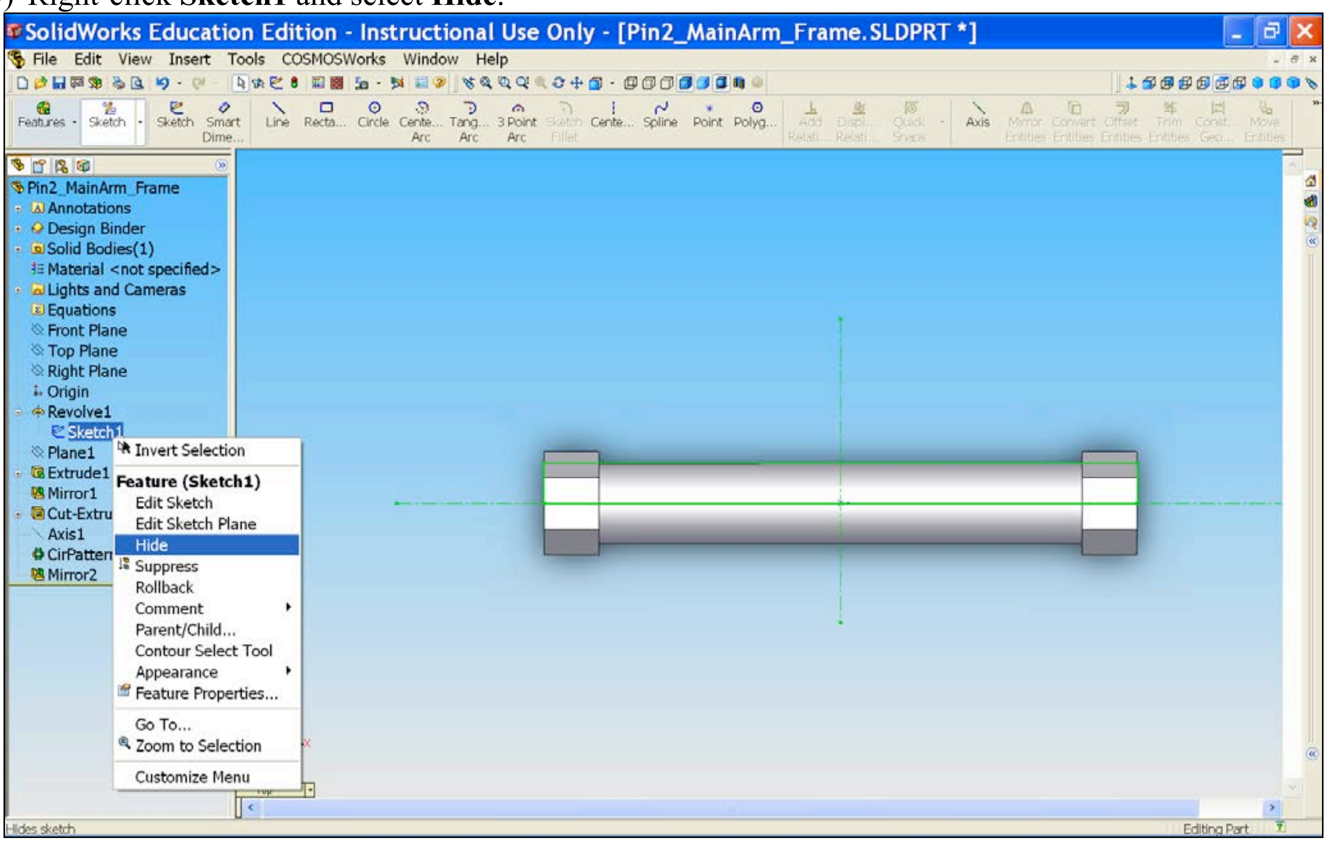
103) The **Modify** box appears. Change the dimension from 13/32" to 1/2".



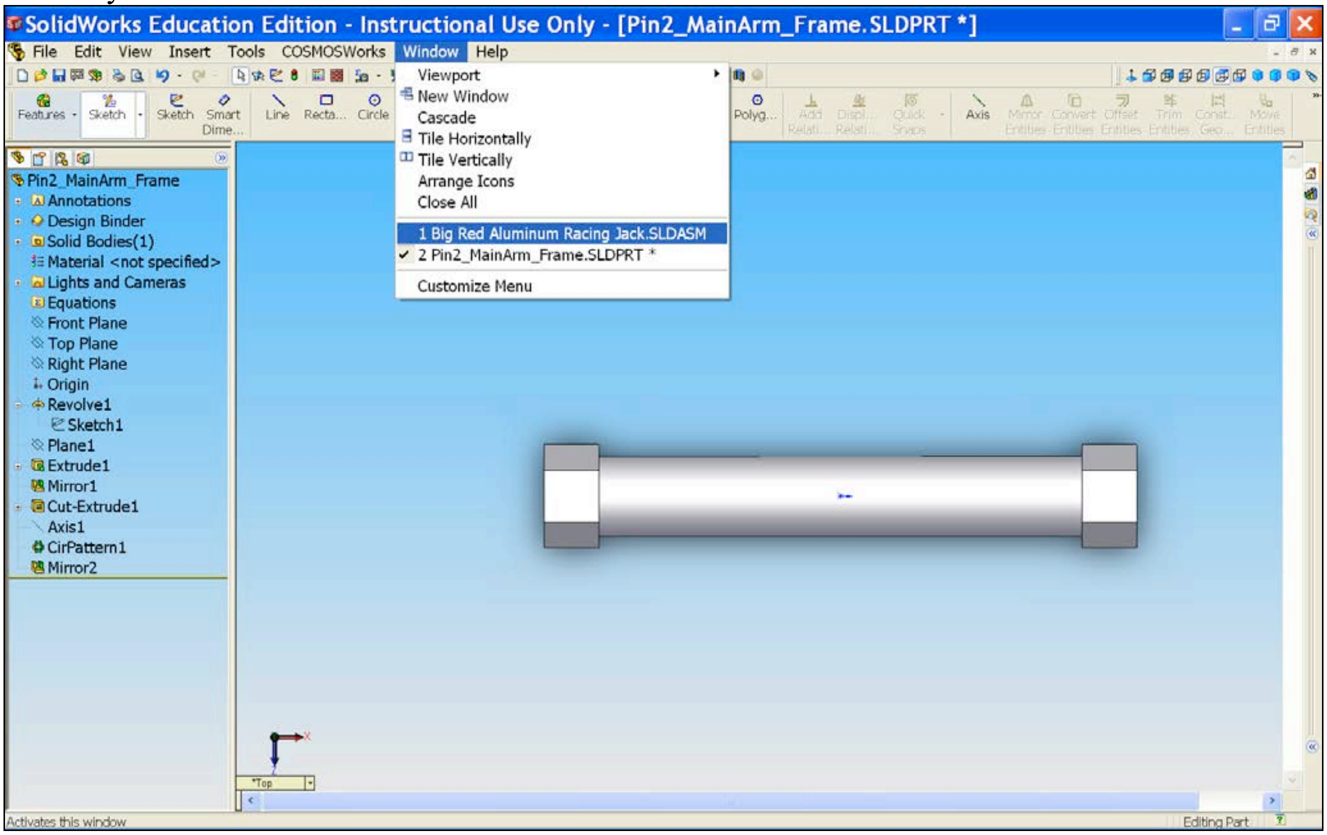
104) Click  **Rebuild** to update the part.



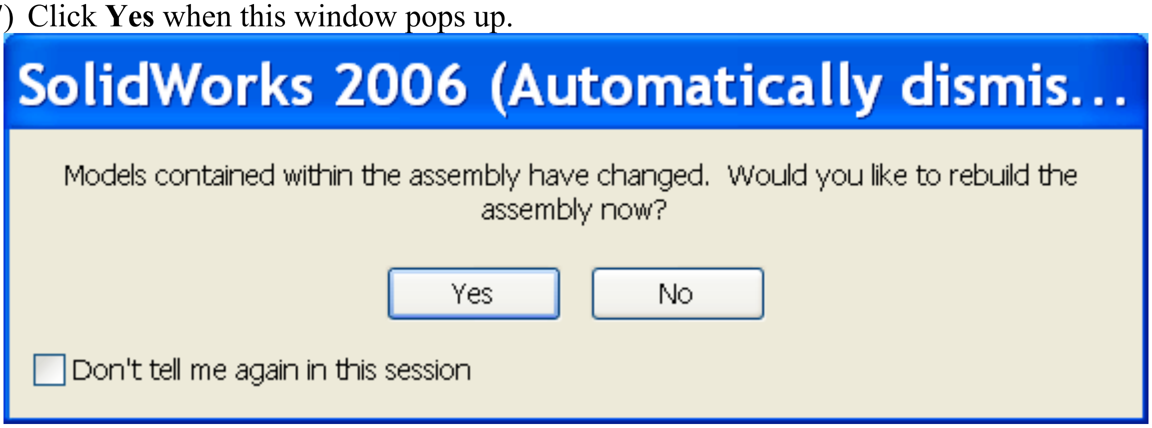
105) Right-click **Sketch1** and select **Hide**.



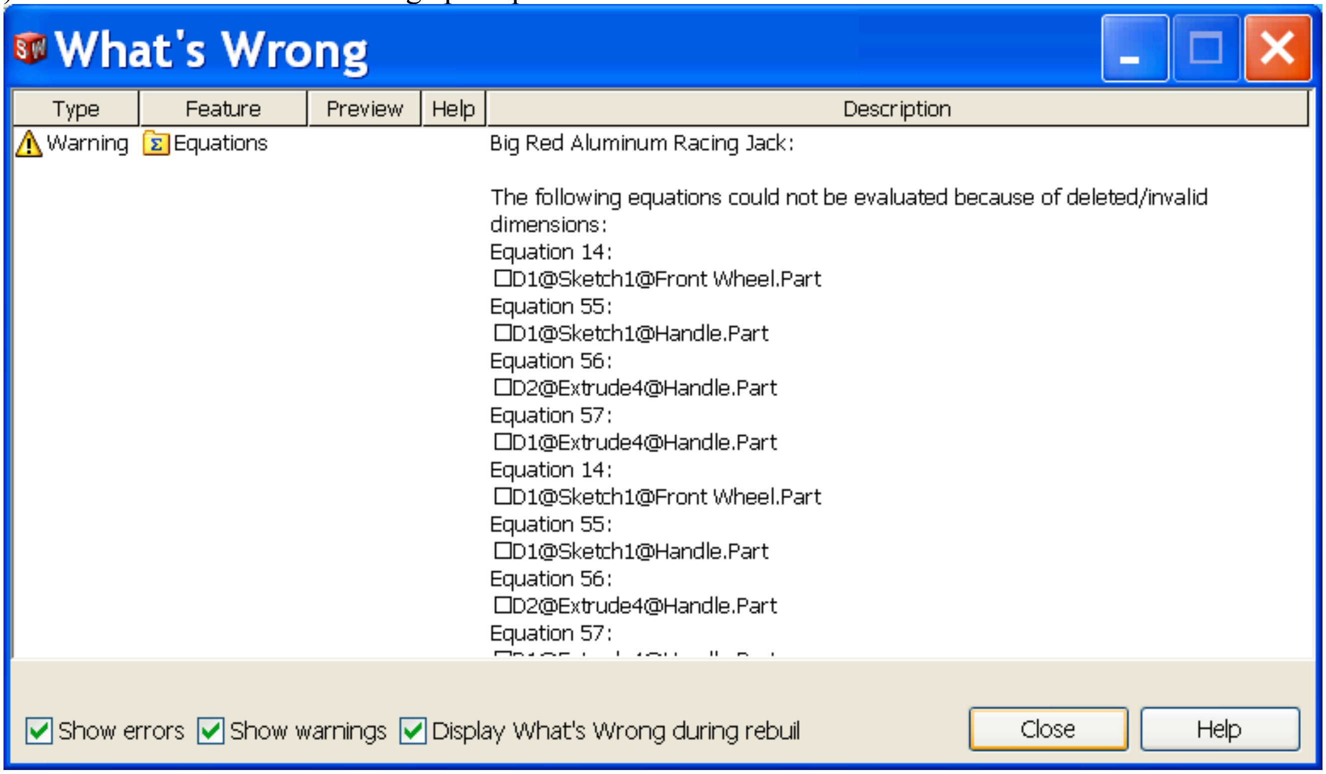
106) Go to **Window** and select **Big Red Aluminum Racing Jack** to go back to the assembly.



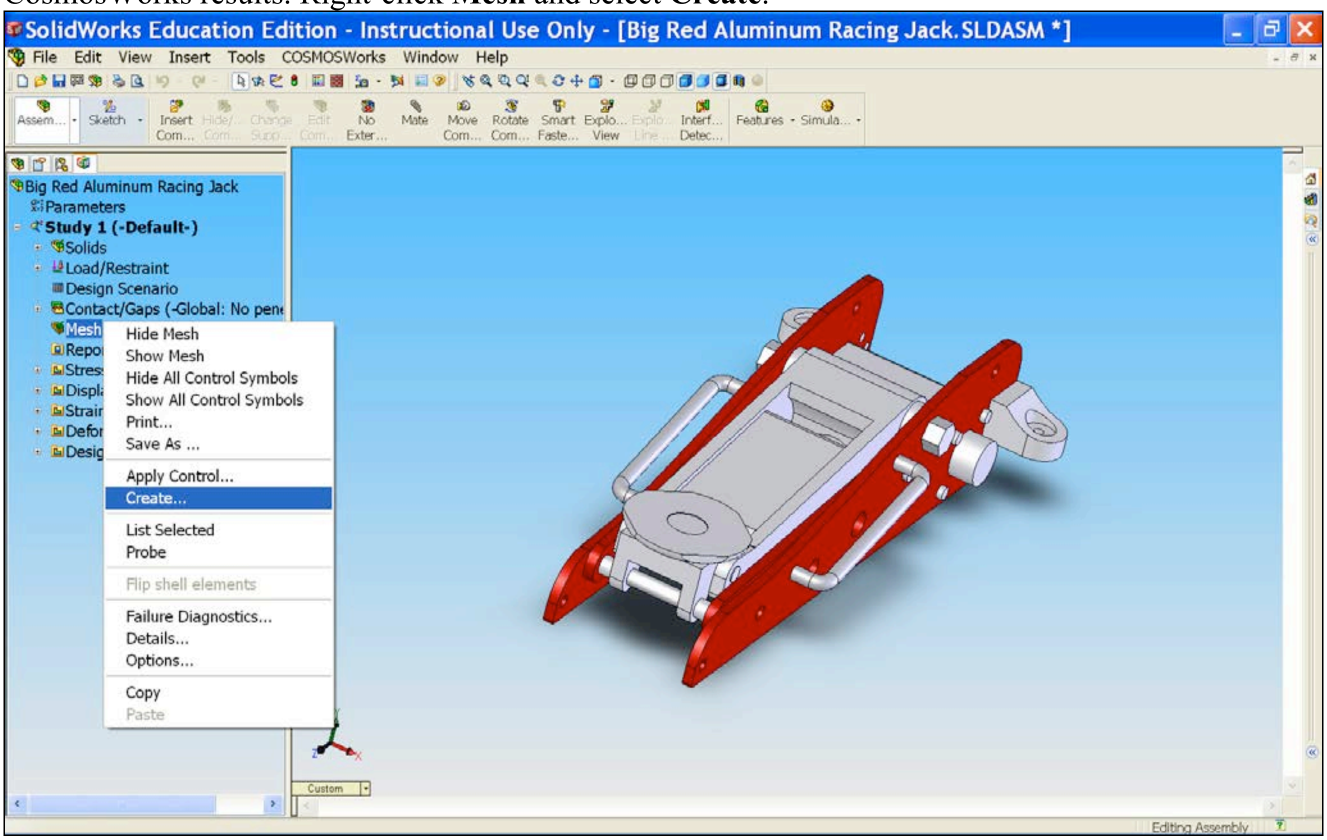
107) Click **Yes** when this window pops up.

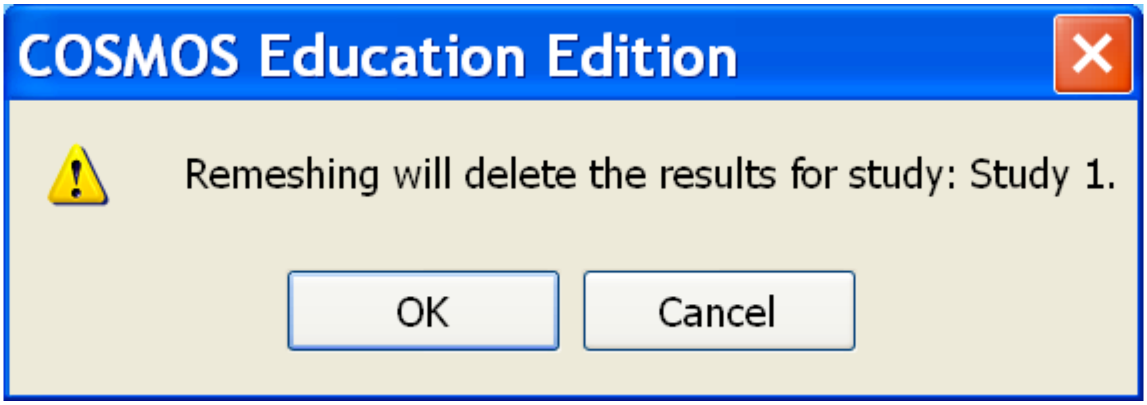


108) Select **Close** when this message prompts.

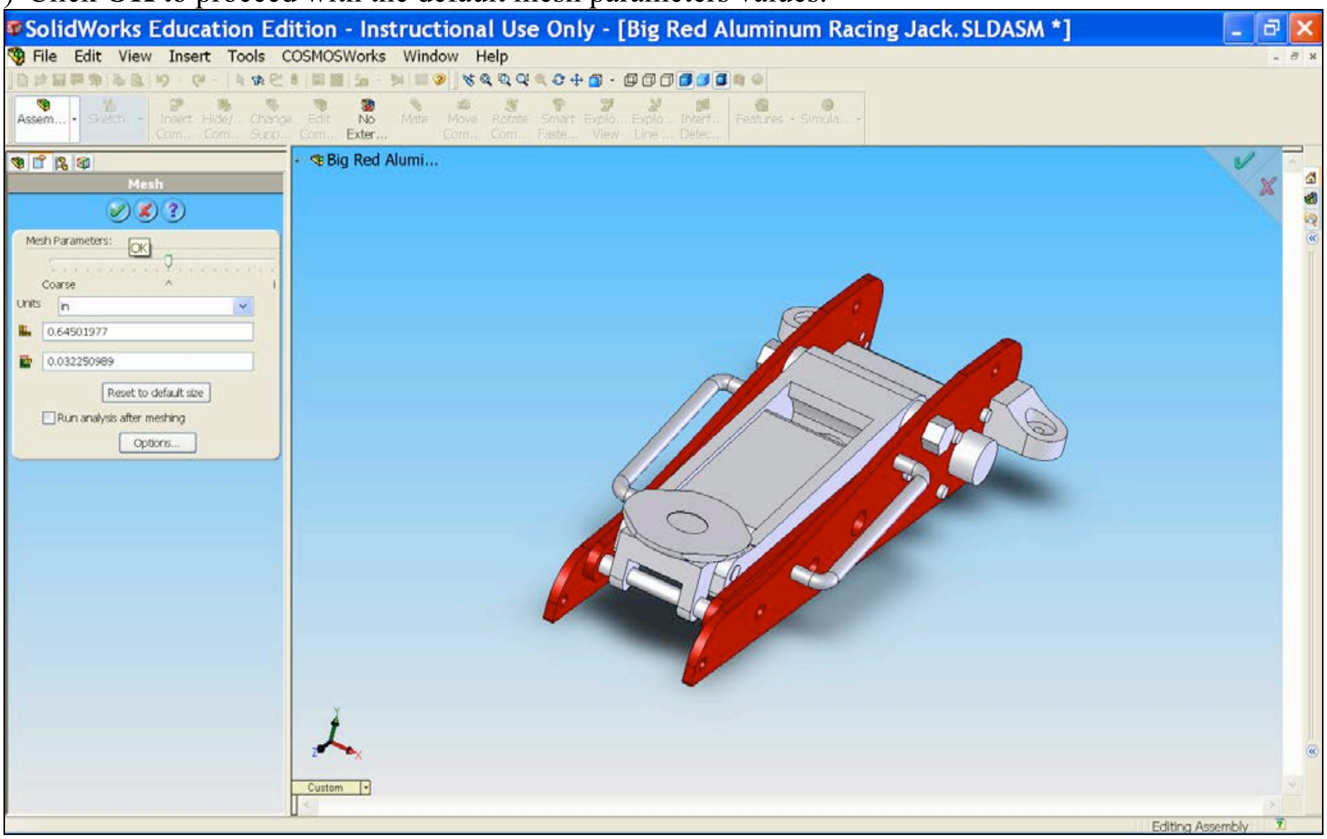


109) Since the dimension of the **Main Arm** was modified, we need to re-create the mesh to update the CosmosWorks results. Right-click **Mesh** and select **Create**.

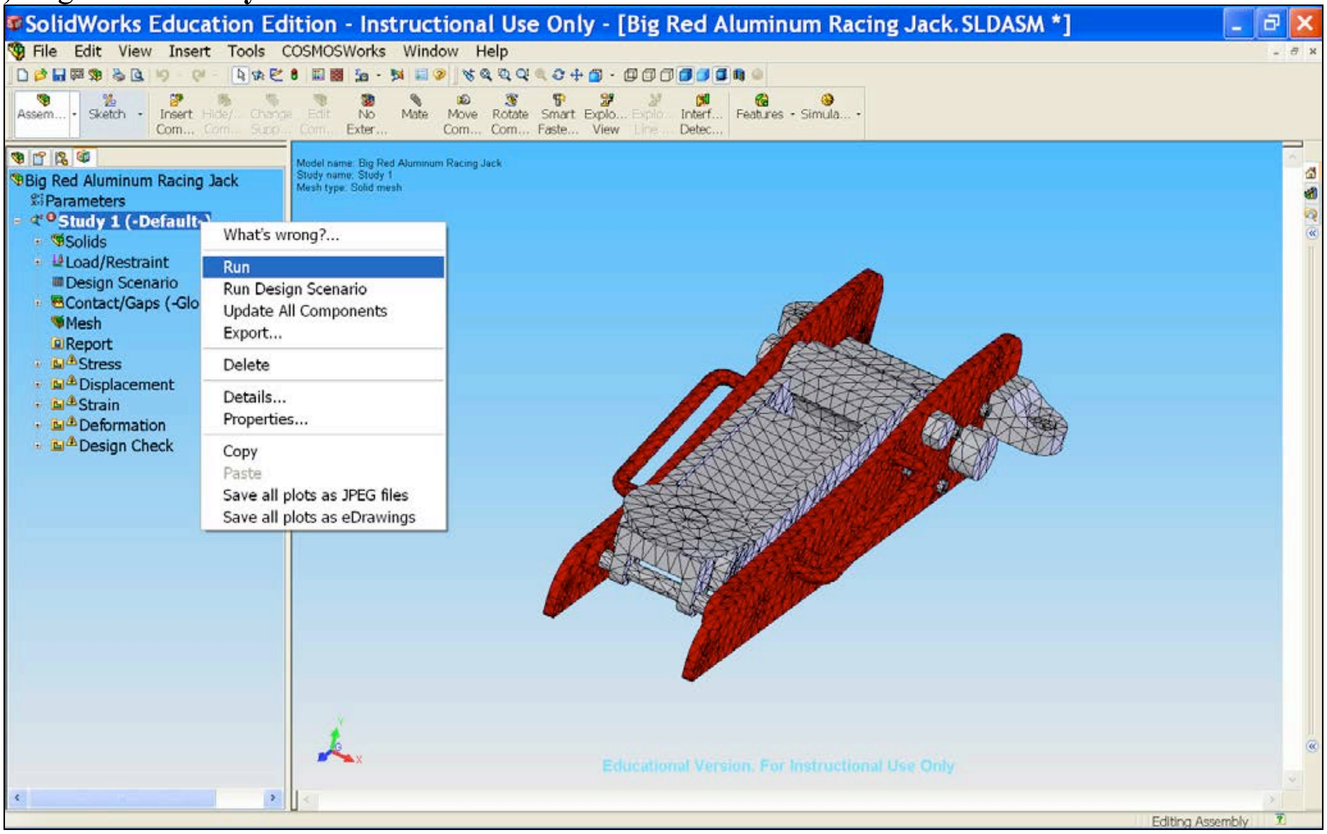


110) 

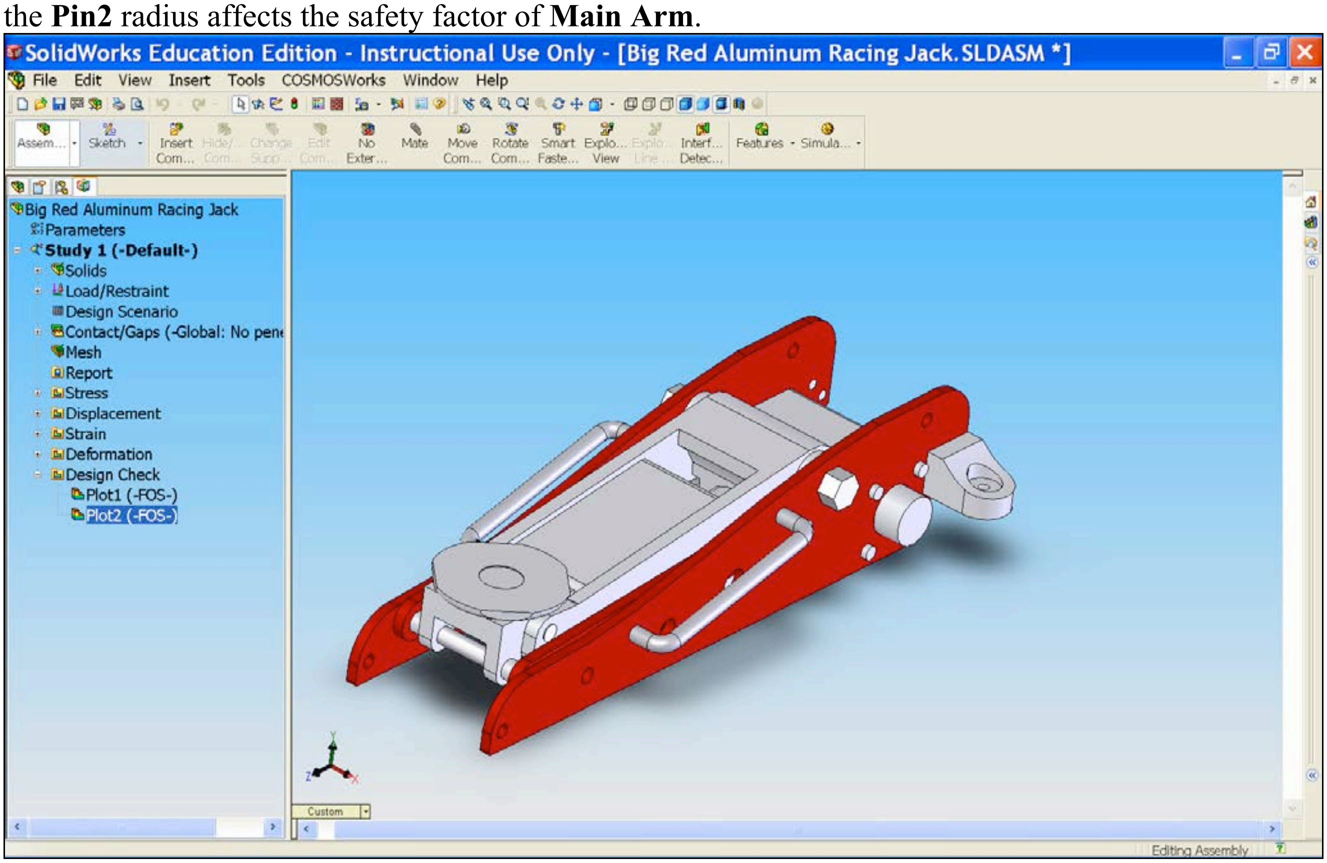
111) Click **OK** to proceed with the default mesh parameters values.



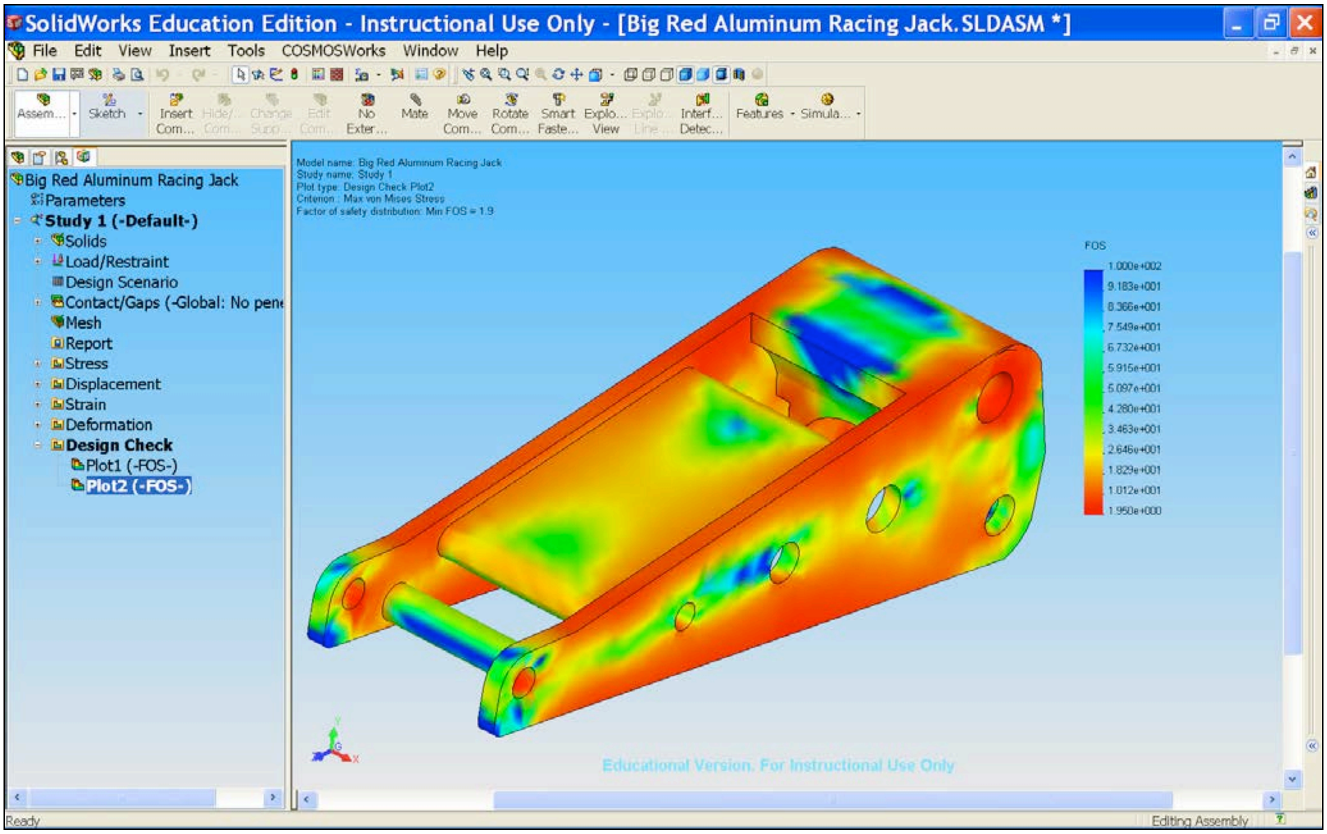
112) Right-click **Study 1** and select **Run**.



113) When CosmosWorks has completed re-running the analysis, double-click **Plot2** to find out how the change in the **Pin2** radius affects the safety factor of **Main Arm**.



114) The safety factor of the **Main Arm** has increased from 1.4 to 1.9 due to the increase of the **Pin2** radius from 13/32" to 1/2".



115) According to the manufacturer of this floor jack, this specific floor jack has a maximum lift of 17 ½" (distance from the **Top Plate** to the ground). In this tutorial, we analyzed the stress on the floor jack when the **Top Plate** is supporting a 3000 lb load at floor jack's minimum lift which is about 4" from the ground. At its maximum lift (17 ½"), the stress throughout the floor jack will be different in comparison to its stress at its minimum lift (4.21"). To find out how the lift height affects the stress throughout the floor jack, you can perform the stress analysis on your own by simply changing the distance between the **Top Plate** and the **Top Plane** as discussed in **Step 7** of this tutorial. You will then need to re-create the mesh and re-run the analysis to update the COSMOSWorks results.