ME 24-688 – Week 13

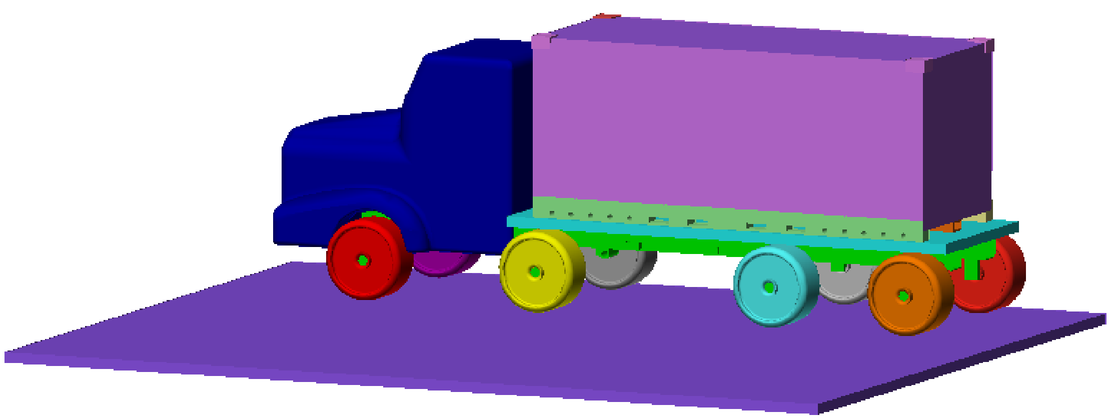
MES Drop Test Simulation

MES Drop Test Simulation

This project will demonstrate Autodesk Simulation's abilities to work with large scale motion, prescribed displacements, and mixed elements even when working with very high element count models. The project will show how to apply prescribed displacements as well as how to optimize contact between parts and surfaces.

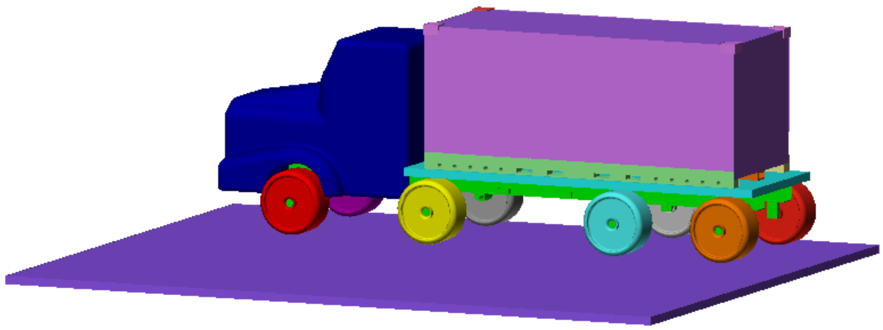
1.1 Project 1 – Drop Test of Container

To simulate a drop test of a container we will setup an analysis using a flatbed truck. The container will start resting on the truck ben then be lifted off by a crane connected to the top corners. Once the container is moving away from the truck bed a problem will accrue causing the container to drop to the ground. The container will make contact with the truck and ground during the fall. This will be completed using a Mechanical Event Simulation within Autodesk Simulation and high speed contact pairs for the quick impact.



1. **Open** the provided *Container Truck.stp* STEP file in Autodesk Simulation. Accept the STEP file default units of inches during the open process and choose **MES with Nonlinear Material Models** as the analysis type.

2. The model is a simplified truck with a loaded container on the flatbed as shown below.



3. When performing a contact analysis in MES with parts that start in contact, it is found to be beneficial to define the contact pairs prior to meshing the model. Defining the contact pairs first will prevent mesh matching that occurs between mated parts during the meshing process. The mismatching of mesh in the contact pairs will improve solution convergence and improve run time.

4. Under the **Selection** tab click **Point** for **Shape** and **Parts** for **Select**. This will allow you to select the complete part.

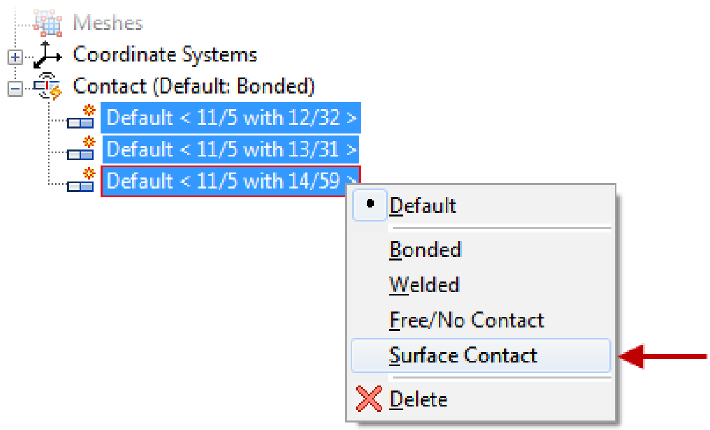
5. Select *Part 11* in the **FEA Editor Browser** and hold down the **<Ctrl>** key and select *Part 12*.

6. Right-Click on one of the parts and select **Contact | Create Contacts between Parts**. This will create a new contact pair between these two entire parts.

7. Repeat steps 5-6 with *Part 11* and *Part 13*.

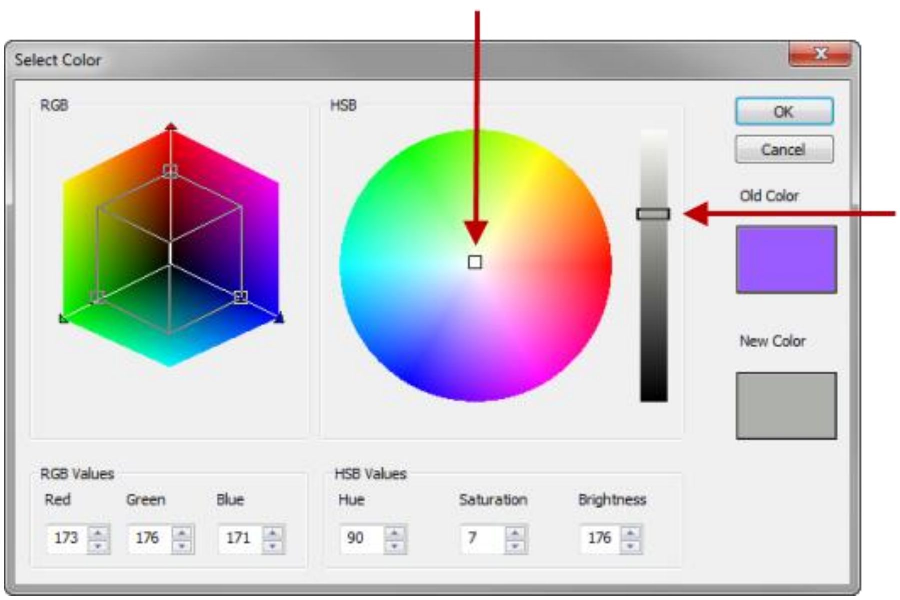
8. Repeat steps 5-6 with *Part 11* and *Part 14*.

9. Select all 3 of the new contacts created in the **FEA Editor Browser** and right-click to set the contact type to **Surface Contact**.



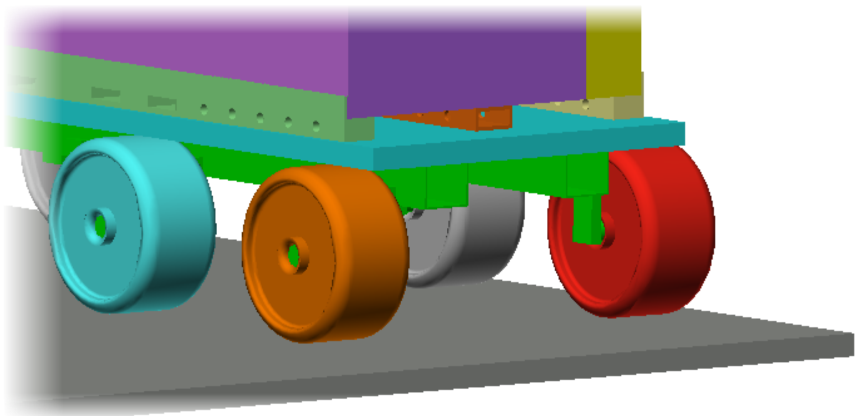
10. Now we will change the color for the ground part to a medium gray. Select the *Part 26* in the **Browser** or select the ground part in the graphics window. Right-Click and select **Edit | Color**.

11. In the **Select Color** dialog drag the **HSB** selector to the center and drag down the slider to a medium gray color. Click **OK** to complete the color assignment.

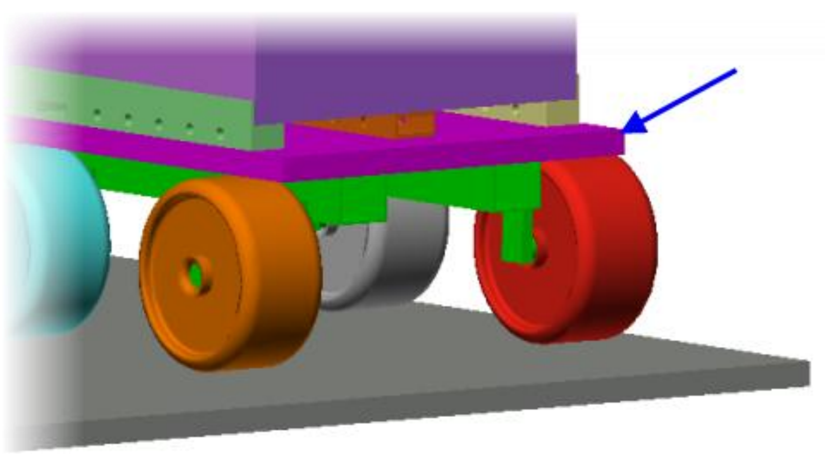


12. Now that we have defined where the container initially rests on the truck bed, we must define where the container might impact when dropped.

13. **Orbit** and **Zoom** the model into a similar viewing position as shown below.

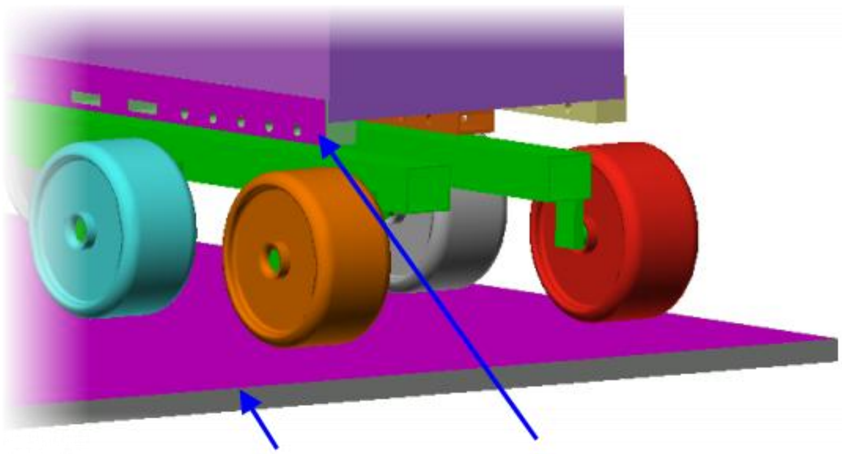


14. Select *Part 11* the flatbed part as marked below. Then right-click and turn off the **Visibility** of the part.



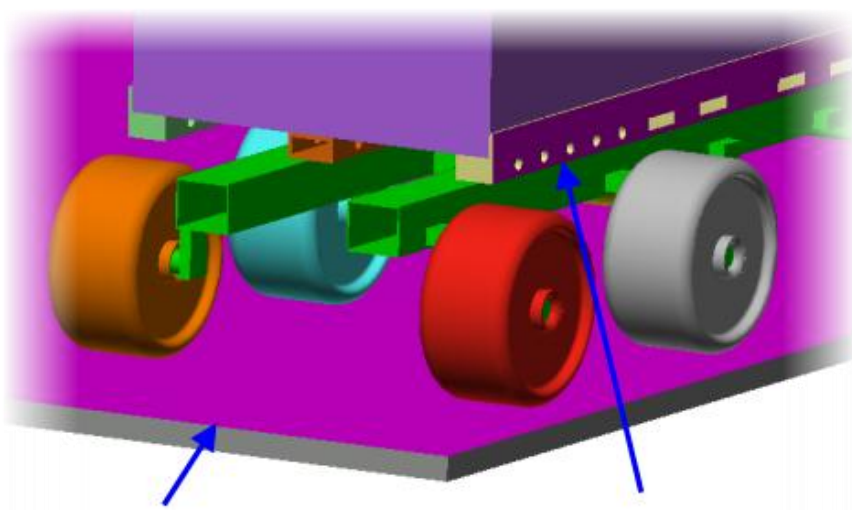
15. Under the **Selection** tab click **Point** for **Shape** and **Surfaces** for **Select**.

16. Select the ground top surface then hold the **<Ctrl>** key and select the side surface of the container lower frame as shown below.

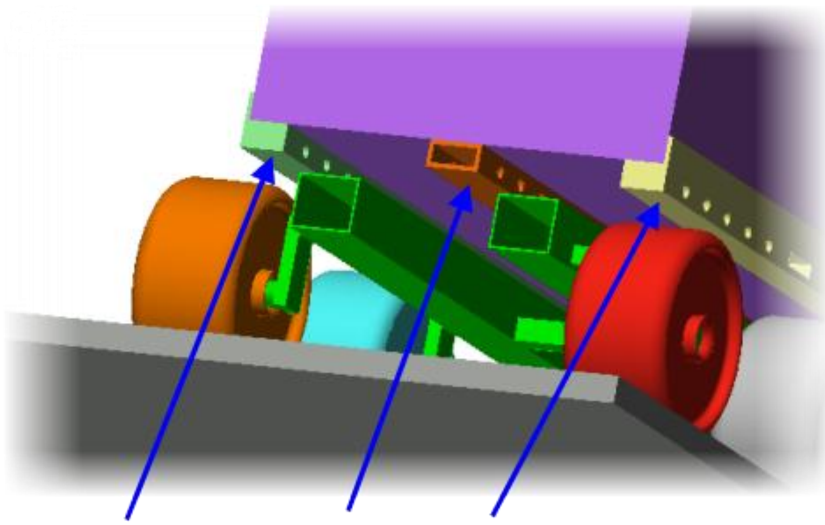


17. Right-Click and select **Contact | Surface Contact**.

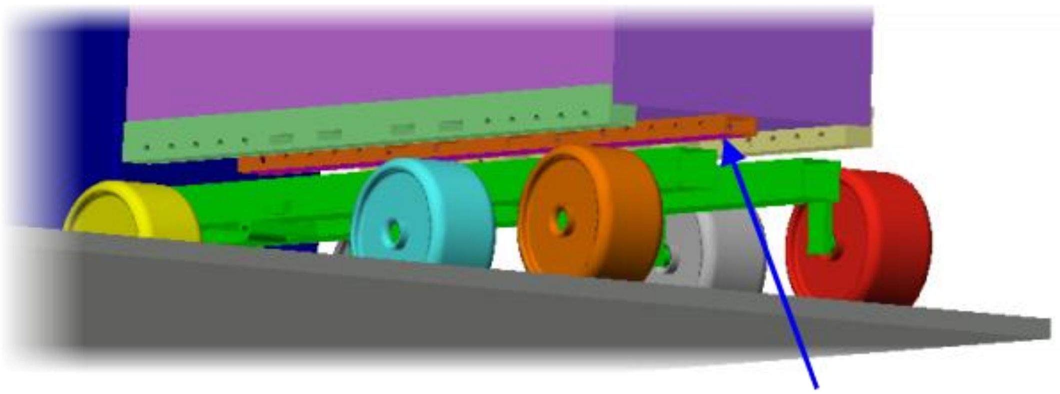
18. Report the process of creating a **Surface Contact** between the ground top surface and other side surface of the container as shown below.



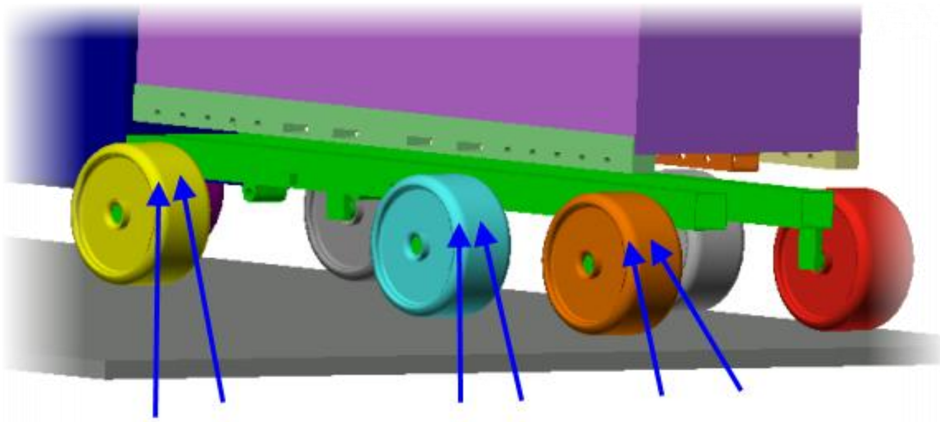
19. Report the **Surface Contact** creation process three more times between the bottom container surfaces as shown below and the ground top surface.



20. Now we will look at where the container if dropped would contact the truck bed and wheels. **Zoom** and **Orbit** into a view similar to the image below. Select the lower surface of the middle container beam.



21. With the container surface already selected create a **Surface Contact** pair between that surface and each of the six (6) wheel surfaces shown below. Tip hold down **<Ctrl>** and deselect one of the wheel surface and then release to select the next surface. This will prevent you from having to reselect the container surface.

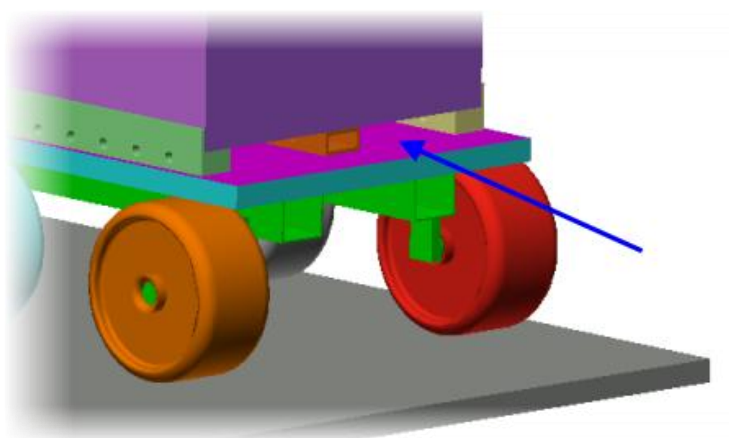


22. At this time there should be fourteen (14) contact pairs in the **FEA Editor Browser**.

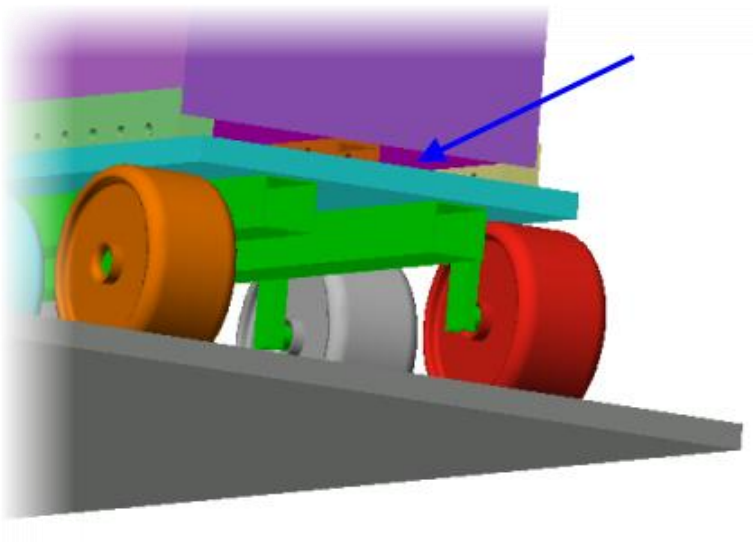
23. To save time during this project only the important contact surfaces have been selected because the outcome is already known.

24. Turn the **Visibility** of *Part 11* the truck bed part.

25. Select the top surface of the truck bed as shown below.



26. **Orbit** the model as required and hold down **<Ctrl>** and select the bottom surface of the container as shown.



27. Rich-Click and select **Contact | Surface Contact** to create a new contact pair between these two surfaces.

28. To prepare the model for meshing click on *Part 19* in the **FEA Editor Browser** and hold **Shift** and click *Part 25*. This will select 7 parts.

29. Right-Click and select **CAD Mesh Options | Part**. The **Part Mesh Settings** dialog will open and choose **Plate/Shell** as the **Mesh Type**.

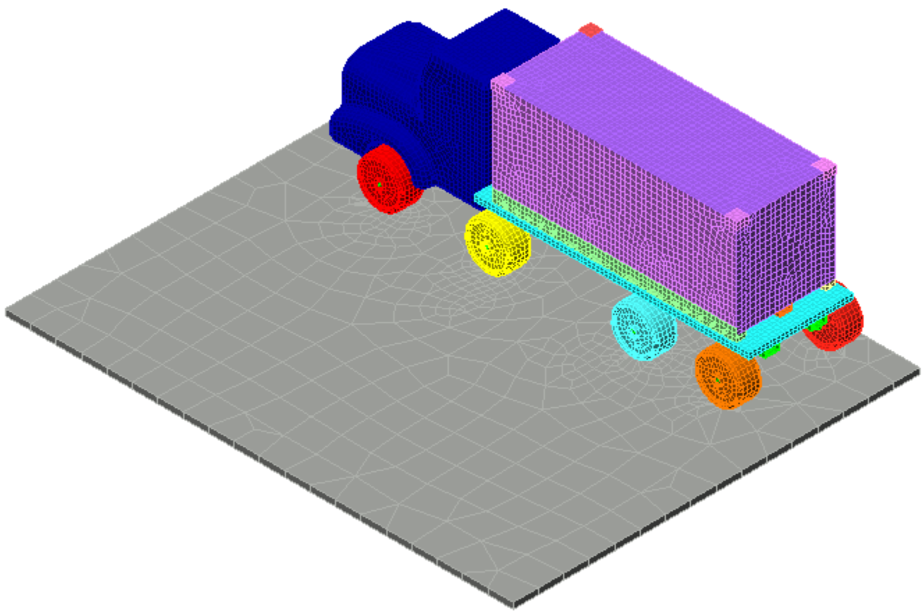
30. Click the **Options** button. Change the **Type** to **Absolute Mesh Size** and enter 5 in for the **Size**. Click **OK** twice to complete. This will assign a specific mesh size to these parts of the overall model.

31. Right-Click on *Part 26* and select **CAD Mesh Options | Part**. Click **Options** and enter an **Absolute Mesh Size** of 25 in and click **OK** twice to complete.

32. Click **Mesh tab | Mesh panel | 3D Mesh Settings**. Click the **Options** button and then select **Model**. Uncheck the checkbox **Use Automatic Geometry Based Mesh Size Function**. Click **OK**.

33. Click the **Mesh Model** button to start the meshing process.

34. Click the **Home** view on the **View Cube**. The meshed model will look like the image below.



35. Now we will define the element information and materials of the parts. Start by changing the **Element Type** of *Part 1* through *Part 10* to use the **3-D Kinematic** element type. Do this by Right-Clicking on the **Element Type** node of each part in the **FEA Editor Browser**.

36. Select *Part 1* through *Part 10* in the **FEA Editor Browser** and Right-Click to select **Edit | Material**. Select the *Steel (ASTM – A36)* material and click **OK** and accept the override to set the material for these 10 parts.

37. Select *Part 11* through *Part 18* in the **FEA Editor Browser** and Right-Click to select **Edit | Material**. Select the *Steel (ASTM – A36)* material and click **OK** and accept the override to set the material for these 8 parts.

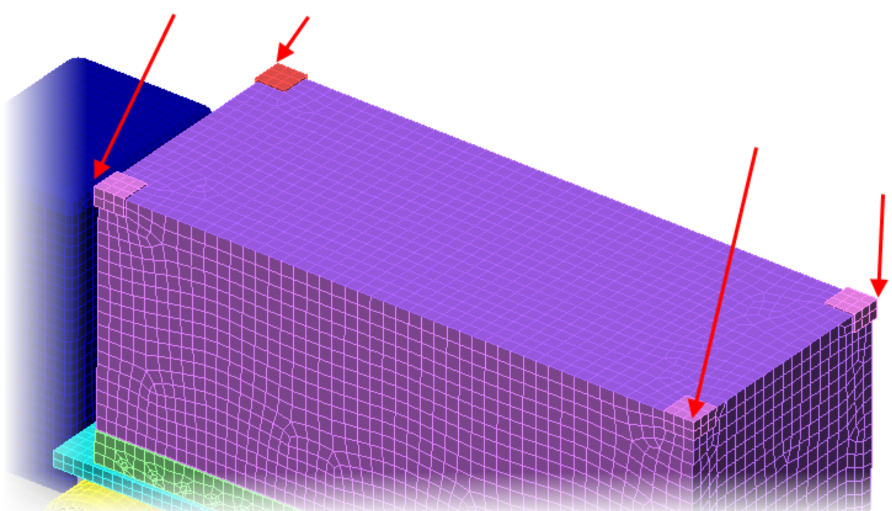
38. Select *Part 19* through *Part 25* in the **FEA Editor Browser** and Right-Click to select **Edit | Element Data**. Enter 0.25 in as the **Thickness** for the shell elements making up the container parts. Click **OK** to complete.

39. Select *Part 19* through *Part 25* in the **FEA Editor Browser** and Right-Click to select **Edit | Material**. Select the *Aluminum 6061-T6; 6061-T651* material and click **OK** and accept the override to set the material for these 6 parts.

40. Right-Click on *Part 26* which is the ground part and selected **Edit | Material**. Select *Concrete (Fairly High Strength)* as the material and click **OK** to complete the material assignment.

41. Under the **Selection** tab click **Point** for **Shape** and **Vertices** for **Select**. This will allow you to select nodes.

42. Select all four outer corner nodes of the top container corner *Parts 15 – 18* as shown below.



43. Click **Setup tab | Constraints panel | Prescribed Displacement** to start the process of assigning movement to the container.

44. Ensure the displacement **Type** is **Translation** and enter -60 as the **Magnitude**. Select **Scalar Y** as the **Direction** to move the container to the side. Change the **Load Case** to 2 and click the **Curve** button.

45. Enter the data below to generate the displacement load curve in the **Multiplier Table Editor** dialog. Click **OK** once complete.

Time (s)	Multiplier
0.0	0.0
2.0	0.0
5.0	1.0
8.0	1.0

46. Click the **Data** button to open the **Active Ranges** interface. Enter 5 in the **Death Time(s)** field and click **OK**. Defining a death time will release the prescribed displacements and allow the container to move freely.

47. Click **OK** to complete the prescribed displacement assignment.

48. Select the same 4 corner nodes on the top of the container. Click **Prescribed Displacement** from the **Constraints** panel.

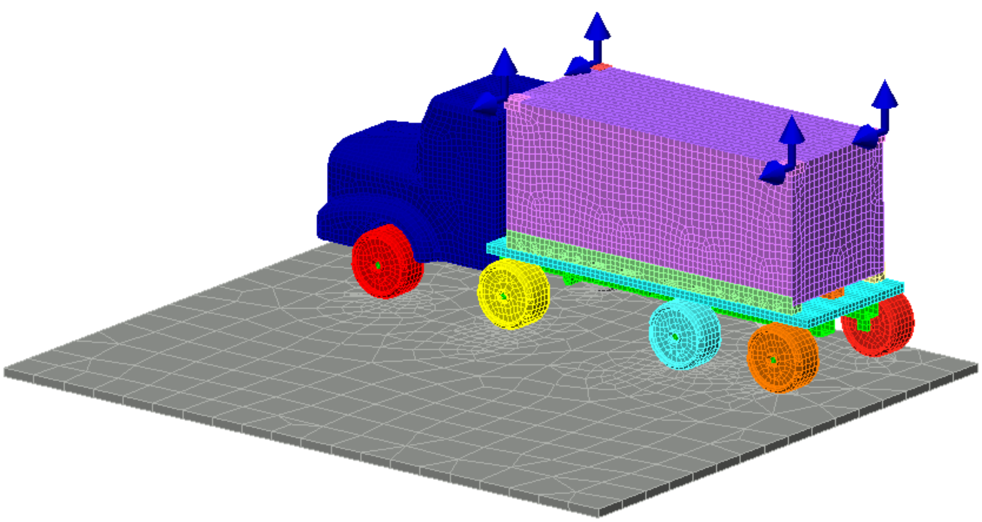
49. Ensure the displacement **Type** is **Translation** and enter 12 as the **Magnitude**. Select **Scalar Z** as the **Direction** to move the container to the side. Change the **Load Case** to 3 and click the **Curve** button.

50. Enter the data below to generate the displacement load curve in the **Multiplier Table Editor** dialog. Click **OK** once complete.

Time (s)	Multiplier
0.0	0.0
2.0	1.0
5.0	1.0
8.0	0.0

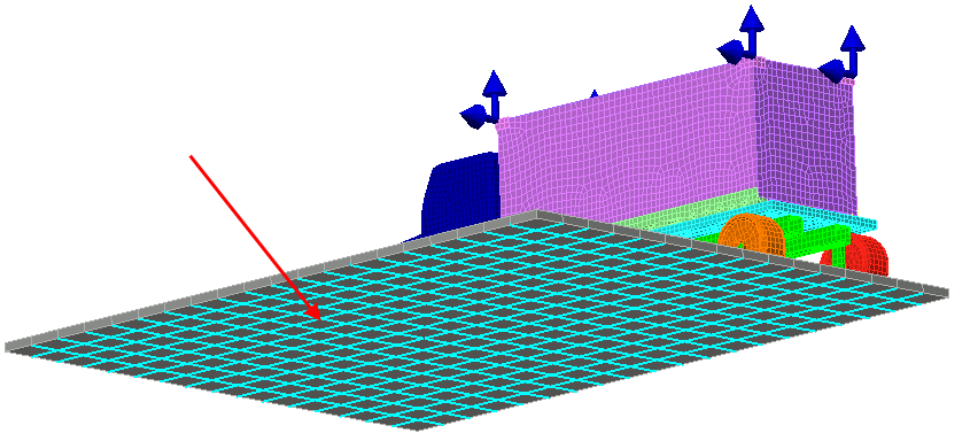
51. Click **OK** to complete the prescribed displacement assignment using the same **Active Range** 1

52. The model should look like the image below showing the two prescribed displacement constraints.



53. Under the **Selection** tab click **Point** for **Shape** and **Surfaces** for **Select**.

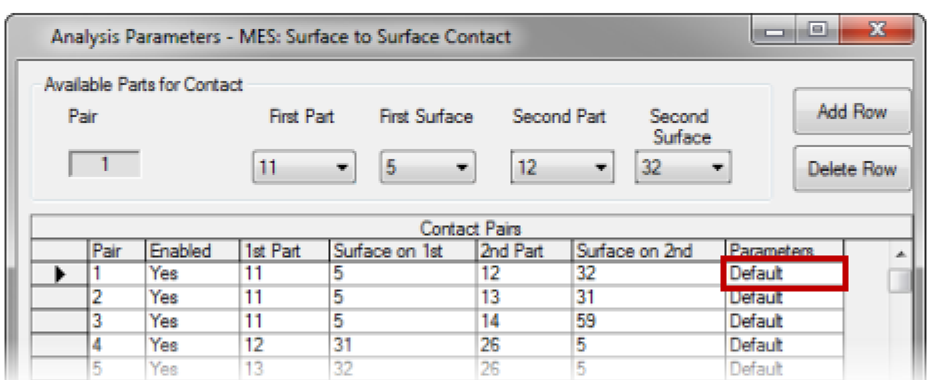
54. Select the bottom surface of the ground part (*Part 26*) as shown below.



55. Click **Setup tab | Constraints panel | General Constraint** then select **Fixed** as the constraint type to constraint all degrees of freedom. Click **OK** to complete.

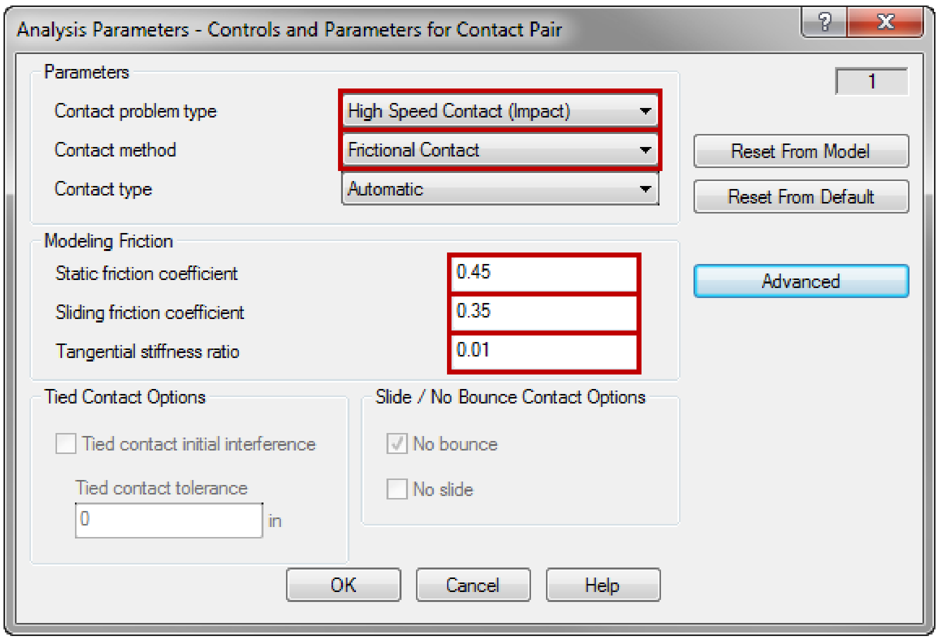
56. In the graphics window Right-Click and select **General Surface-to-Surface Contact**. This will open the contact pairs dialog.

57. Within the dialog click on the **Default** cell of the first contact pair row as shown below.



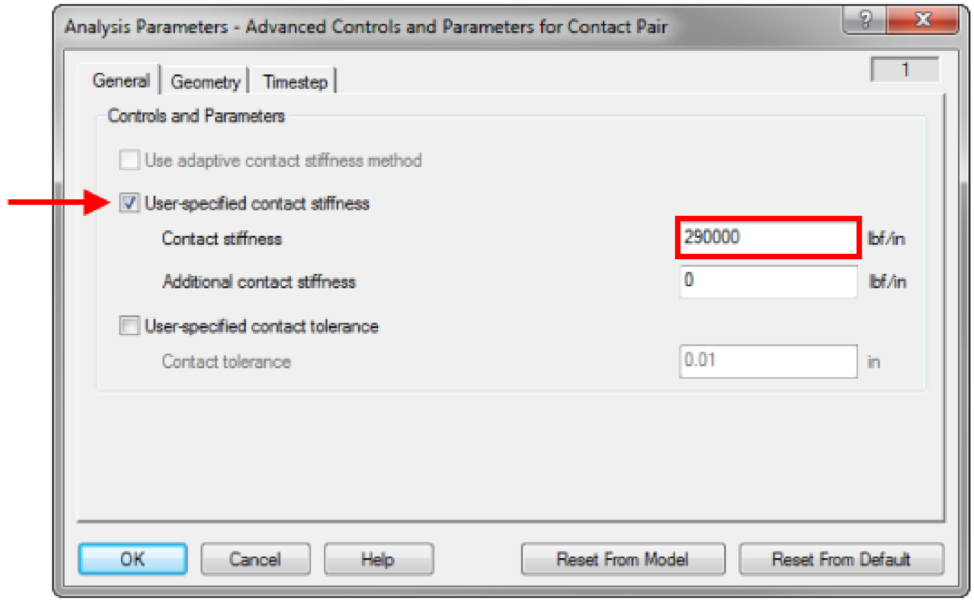
58. Complete the following items:

- Change the **Contact Problem Type** to **High Speed Contact (Impact)**
- Change the **Contact Method** to **Frictional Contact**
- Change the **Static Friction Coefficient** to 0.45
- Change the **Sliding Friction Coefficient** to 0.35
- Change the **Tangential Stiffness Ratio** to 0.01



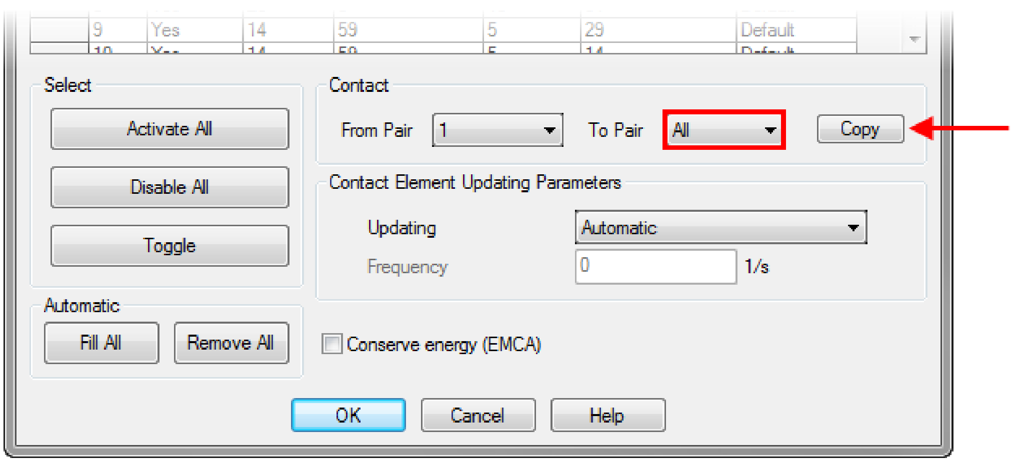
59. Click the **Advanced** button.

60. Check the **User-Specified Contact Stiffness** checkbox. Enter 290000 for the **Contact Stiffness**. Click **OK** to complete.

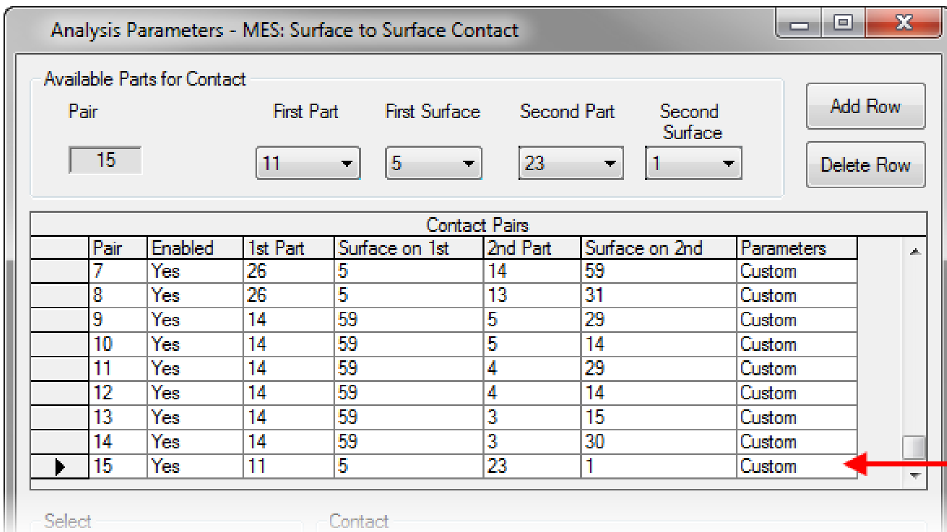


61. Click **OK** to exit editing the first constraint pair.

62. Select **All** for the **To Pair** field under **Contact**. Then click the **Copy** button and select **Yes** to copy the settings established for the first pair to all pairs.



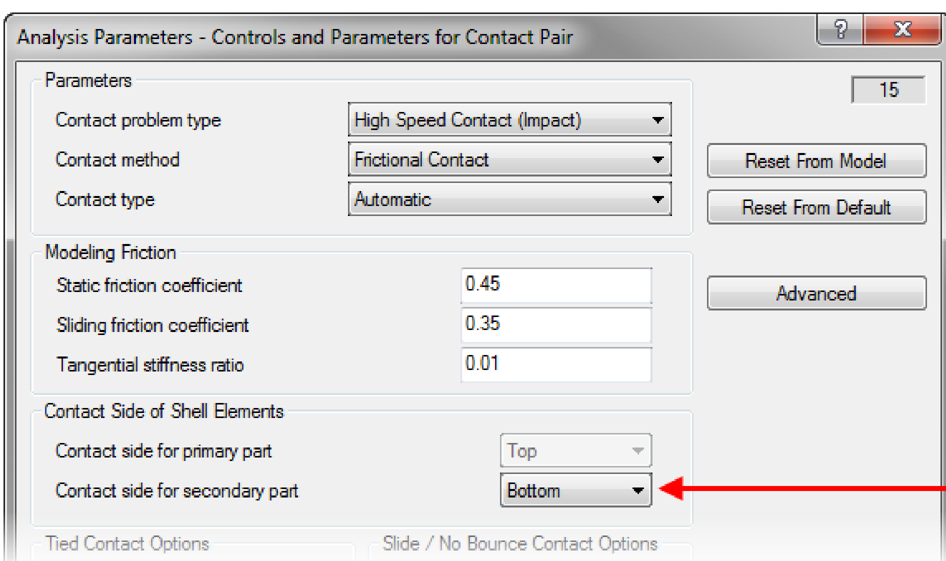
63. Click the word **Custom** in the **Parameters** column for the contact pair defined between the bottom of the container to the top of the truck bed. This contact pair should be between **Part 11 / Surface 5** and **Part 23 / Surface 1**.



Pair	Enabled	1st Part	Surface on 1st	2nd Part	Surface on 2nd	Parameters
7	Yes	26	5	14	59	Custom
8	Yes	26	5	13	31	Custom
9	Yes	14	59	5	29	Custom
10	Yes	14	59	5	14	Custom
11	Yes	14	59	4	29	Custom
12	Yes	14	59	4	14	Custom
13	Yes	14	59	3	15	Custom
14	Yes	14	59	3	30	Custom
15	Yes	11	5	23	1	Custom

64. Change the **Contact Side of Shell Elements** to **Bottom**. Click **OK** to complete.

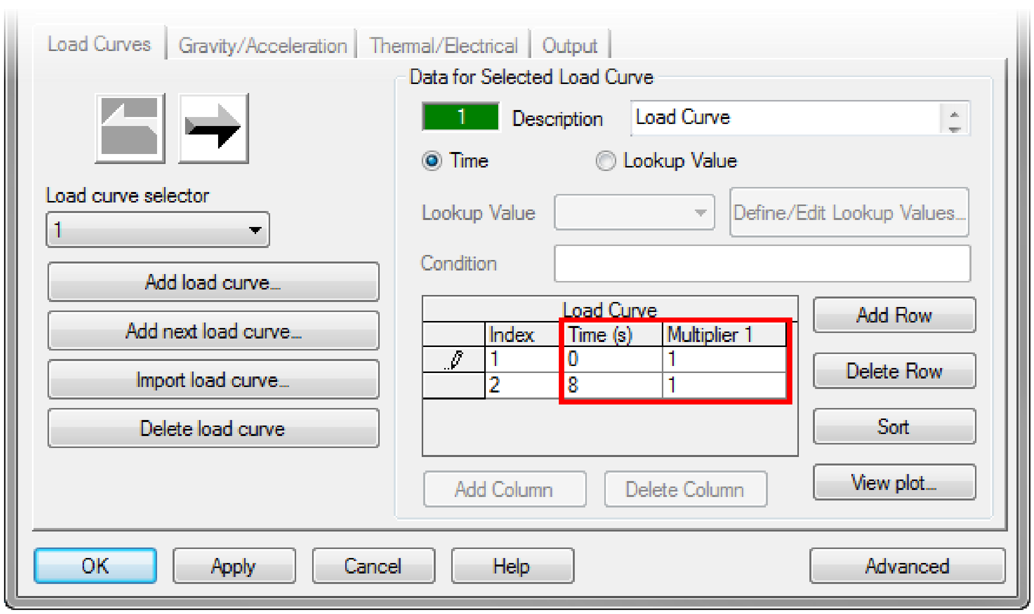
Note: Whether the shell part is defined as Primary or Secondary will depend on the order of selection when defining the contact pair.



65. Click **OK** to complete the editing of the contact pairs.

66. Click **Setup tab | Model Setup panel | Parameters** to **Analysis Parameters** dialog. Enter 8 seconds for the **Duration** and verify the **Capture Rate** is 20.

67. Ensure **Load Curve 1** is selected. Edit the **Time** to have 8 seconds in the second row and verify 0 in the first row. Then enter 1 for the **Multiplier** in both rows as shown below. Click **Apply** to complete.



68. Choose the **Gravity / Acceleration** tab. Click the **Set for Standard Gravity** button to populate the standard value for gravity. Ensure that the **Z Multiplier** is set to -1. Click **OK** to complete.

69. **Save** the model.

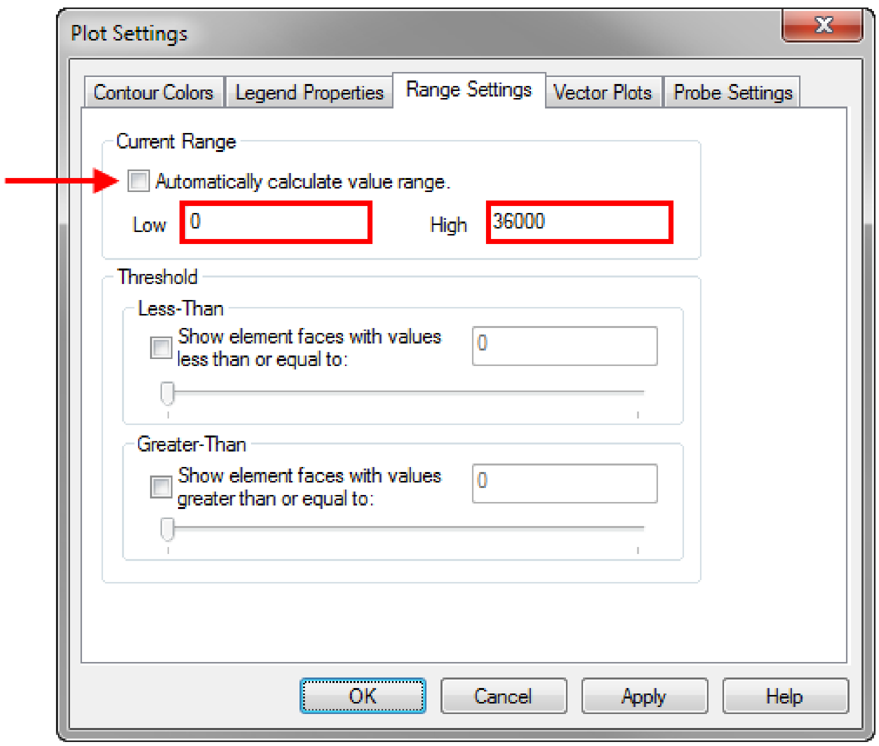
70. The model is now ready to analyze. From the **Analysis** tab click **Run Simulation**.

    Note: The meshing process will take a couple of minutes to complete and the analysis will take about 5 hours to complete depending on computer resources. There is a completed analysis file provided that you may open instead of solving named *flatbed.fem*.

71. If you open the provided completed analysis file click the **Results** tab in the **Browser**.

72. To make identifying areas of possible failure, change the legend properties to range from 0 to 36,000, where 36,000 is the lowest yield strength in the assembly. This can be completed in the **Results Contours tab | Settings panel | Legend Properties – Setup**.

73. Select the **Range Settings** tab and uncheck the **Automatically Calculate Value Range** checkbox. Enter 0 for the **Low** and 36000 for the **High** value. Click **OK** to complete.



74. Use the Load Case forward and back buttons to play with viewing each of the analysis time steps.

75. To find the time step / load case with the maximum stress, go to **Results Inquire tab | Inquire panel | Maximum Results Summary**. This will display the maximum stress for each time step. You can identify which time step has the highest stress then use the **Set Load Case** feature to view the results of that time step.