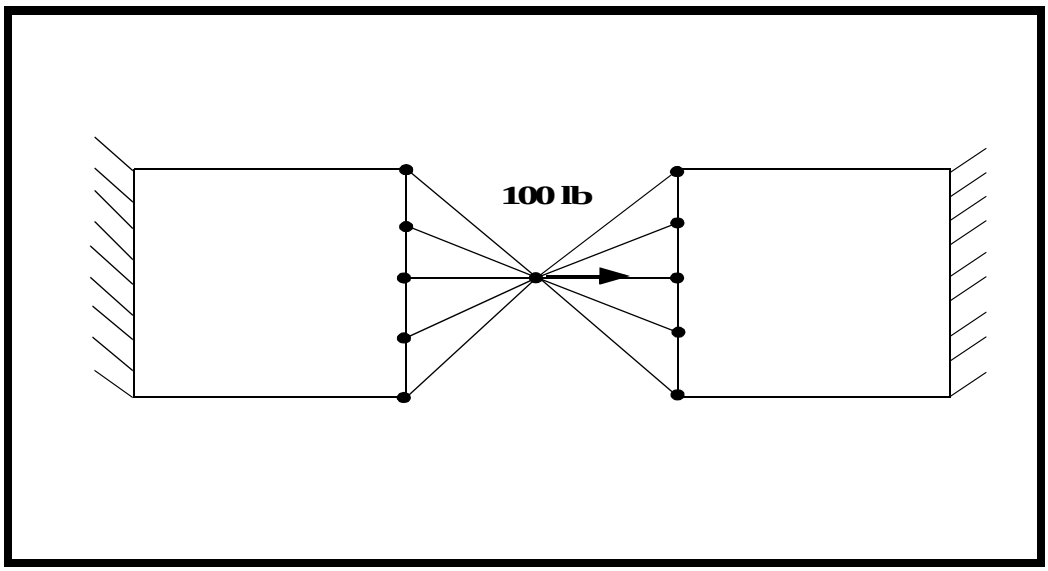
WORKSHOP 34

RBE2 vs. RBE3



Objectives:

- Demonstrate the difference between two rigid body elements.
- Run a linear static analysis.
- Create a deformation plot of the model.
- Review the results.

Model Description:

The figure below shows two identical plates. One edge of each plate is fixed while a total load of 100 lbs is applied horizontally at the node between the two plates. The load will be distributed to the plates using rigid elements. In part A of this exercise, the rigid element is an RBE2 (true rigid element). In part B, the rigid element is an RBE3 (interpolation element). Figure 34.1 displays the schematic of two identical plates. Table 34.1 below displays all the necessary properties of the model.

Figure 34.1 - Schematic of two identical plates.

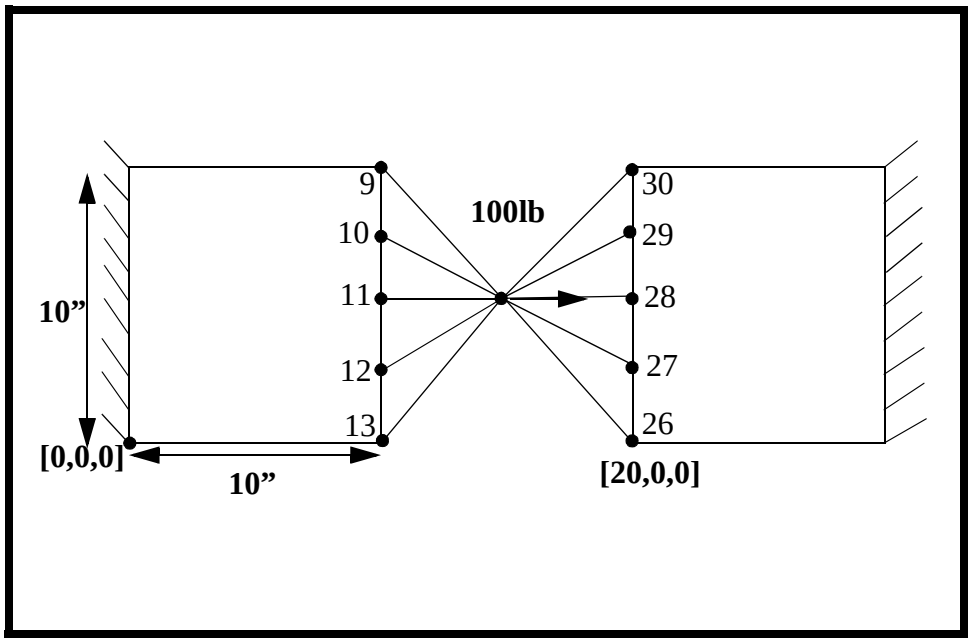


Table 34.1 -Plate Properties.

Area of each plate:	**100 sq. in.**
Youngs Modulus:	**10E+06 psi**
Poisson Ratio:	**0.3**
Applied Force:	**100 lb**

Exercise Procedure:

1. Start up MSC.Nastran for Windows and create a new model.

   Start MSC.N4W by double-clicking on the MSC.N4W icon. When the Open Model File dialog box appears; choose New Model.

Open Model File:	**New Model**

2. Define a material for the model.

   Model/Material...

Title:	**mat_1**
Youngs Modulus, E:	**10e6**
Poisson Ratio:	**0.3**

   OK

   Cancel

3. Create the geometry of the two plates.

   Geometry/Surface/Corners...

   Enter the following 8 points.

X:	**0**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**10**	*Z:*	**0**	**OK**
X:	**10**	*Y:*	**10**	*Z:*	**0**	**OK**
X:	**10**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**20**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**20**	*Y:*	**10**	*Z:*	**0**	**OK**
X:	**30**	*Y:*	**10**	*Z:*	**0**	**OK**
X:	**30**	*Y:*	**0**	*Z:*	**0**	**OK**

   Cancel

Turn off the workplane. Right click on screen.

Workplane...

Uncheck **Draw Workplane**.

☐ **Draw Workplane**

Done

View/Autoscale <Ctrl+A>

4. Now create a property for the plates.

Model/Property...

Elem/Property Type...

Plane Elements: ● **Plate**

OK

Title: **prop_1**

Material: **1..mat 1**

Thicknesses: **0.2**

OK

Cancel

5. Create finite elements for the plates.

Mesh/Mesh Control/Size On Surface...

Select All

OK

Element Size: **2.5**

OK

Cancel

6. We are now ready to mesh the plates.

Mesh/Geometry/Surface...

Select All

OK

Property: **1..prop 1**

OK

7. Create a node between the two plates.

Model/Node...

X: **15** *Y:* **5** *Z:* **0** **OK**

Cancel

8. Now create the **RBE2** element. The middle node will be the independent node while the plate edge nodes will be dependent.

Model/Element...

Type...

Other Elements: ● **Rigid**

OK

Click on the node that was just created.

Independent/Node: **51**

Nodes...

Method^ **on Curve**

Select the nodes on the **right edge of the left plate** and the **left edge of the right plate**.

OK

OK

Cancel

9. We are now ready to apply constraints to the plates.

Model/Constraint/Nodal...

Title: **fixed**

OK

Method^ **on Curve**

Select the nodes on the **left edge of the left plate** and the **right edge of the right plate**.

OK

Fixed

OK

Cancel

10. Now create the load.

Model/Load/Nodal...

Title: **load**

OK

Click on the node between the plates.

OK

(highlight) **Force**

☒ ***FX:*** **100**

OK

Cancel

11. Analyze the model and review the results.

File/Export/Analysis Model...

Type: **1..Static**

File name: **rigid_body**

Write

Additional Info: ☒ **Run Analysis**

Advanced...

OK

Output Requests: **2..Print and Postprocess**

☒ **MPC Force**

OK

OK

Yes

File name: **rigid_body**

Save

When the MSC.Nastran Manager is through running, MSC.Nastran will be restored on your screen.

"OK to read plate element corner stresses?"

Yes

The *Message Review* form will appear. If the analysis ran successfully without any fatal errors, continue.

Continue

12. After the analysis, MSC.N4W automatically reads in the output results, which are available for a wide array of graphical and numerical postprocessing.

View/Select... <F5>

Deformed Style: ● **Deform**

Deformed and Contour Data...

Output Set: *(pulldown)* **1..MSC/NASTRAN Case 1**

Deformation: *(pulldown)* **1..Total Translation**

OK

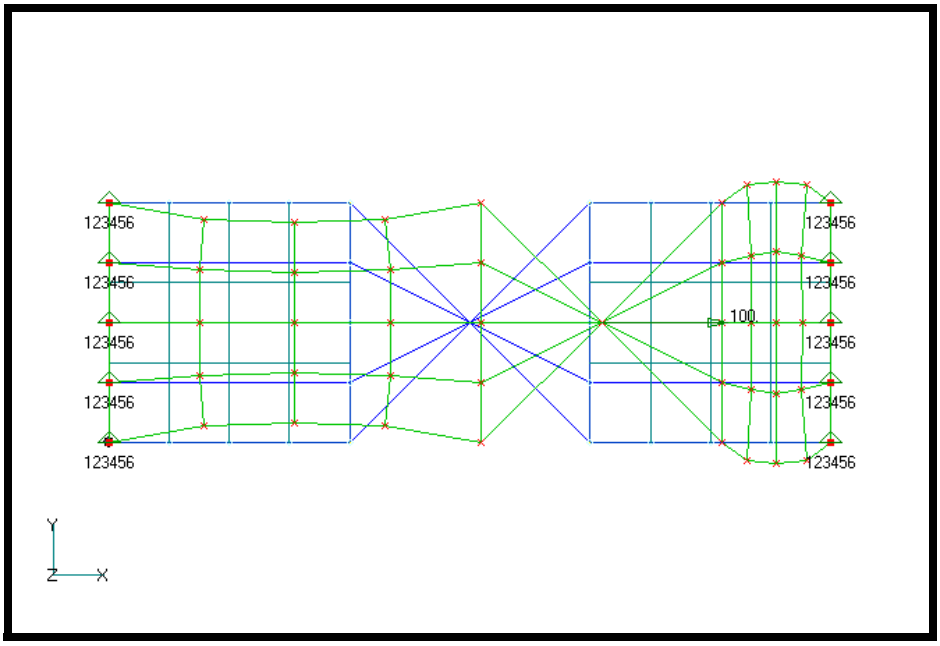
OK

Turn on the *Entity Query* feature to view Node information. Click on the **Off** button in the bottom right hand corner of the screen, and change it to:

Node...

Place the cursor on any of the nodes on the plate edges to see the displacement results. Notice that the edge nodes connected to the RBE2 do not move in the y-direction. These nodes are the dependent nodes on the RBE2, and because of this, they cannot move relative to each other.

Figure 34.2 - Deformed plot for Case 1.



13. To view the MPC forces.

Quick Options... **<Ctrl+Q>**

All Entities Off

(turn on) ☒ **Element**

(turn on) ☒ **Node**

Done

View/Select... **<F5>**

Deformed Style: ● **Vector**

Deformed and Contour Data...

Deformation: **151..Total MultiPointForce**

OK

OK

To view the actual numbers:

View/Options... **<F6>**

Category: ● **PostProcessing**

Options: **Vector Style**

Label Mode: **1..Output Value**

Component Mode **1..Component Vectors**

OK

Zoom into a node to better see the vectors, or move your mouse over the node of interest. Compare the results to the following table.

The nodal results read from the *Entity Query* are as follows:

	MPC Forces (set to 151..Total MultiPoint Force)		**Displacement** (set to 1..Total Translation)	
Node	**T1**	**T2**	**X**	**Y**
9	**7.27**	**2.02**	**2.42e5**	**0**
10	**11.90**	**1.03**	**2.42e5**	**0**
11	**11.66**	**0**	**2.42e5**	**0**
12	**11.90**	**-1.03**	**2.42e5**	**0**
13	**7.27**	**-2.02**	**2.42e5**	**0**
30	**7.27**	**-2.02**	**2.42e5**	**0**
29	**11.90**	**-1.03**	**2.42e5**	**0**
28	**11.66**	**0**	**2.42e5**	**0**
27	**11.90**	**1.03**	**2.42e5**	**0**
26	**7.27**	**2.02**	**2.42e5**	**0**

14. Now we will create the **RBE3** element. The middle node will be the reference node, and the nodes on the edge of the plates will be the nodes to average.

View/Select... <F5>

Deformed Style: ● **None**

OK

Delete/Model/Element...

Click on the element between the two plates.

OK

Yes

View/Autoscale <Ctrl+A>

Model/Element...

Interpolation: ☒ **Factor** **1**

DOF: ☒ **TX** ☒ **RX**
☒ **TY** ☒ **RY**
☒ **TZ** ☒ **RZ**

Reference/Node: **<select center node>**

DOF: ☒ **TX** ☒ **RX**
☒ **TY** ☒ **RY**
☒ **TZ** ☒ **RZ**

Nodes...

Method^ **on Curve**

Select the nodes on the **right side of the left plate** and the **left side of the right plate**.

OK

OK

Cancel

15. Repeat **Step 11** to analyze the model.

16. After the analysis, MSC.N4W automatically reads in the output results, which are available for a wide array of graphical and numerical postprocessing.

View/Select... <F5>

Deformed Style: ● **Deform**

Deformed and Contour Data...

Output Set: *(pulldown)* **2..MSC/NASTRAN Case 1**

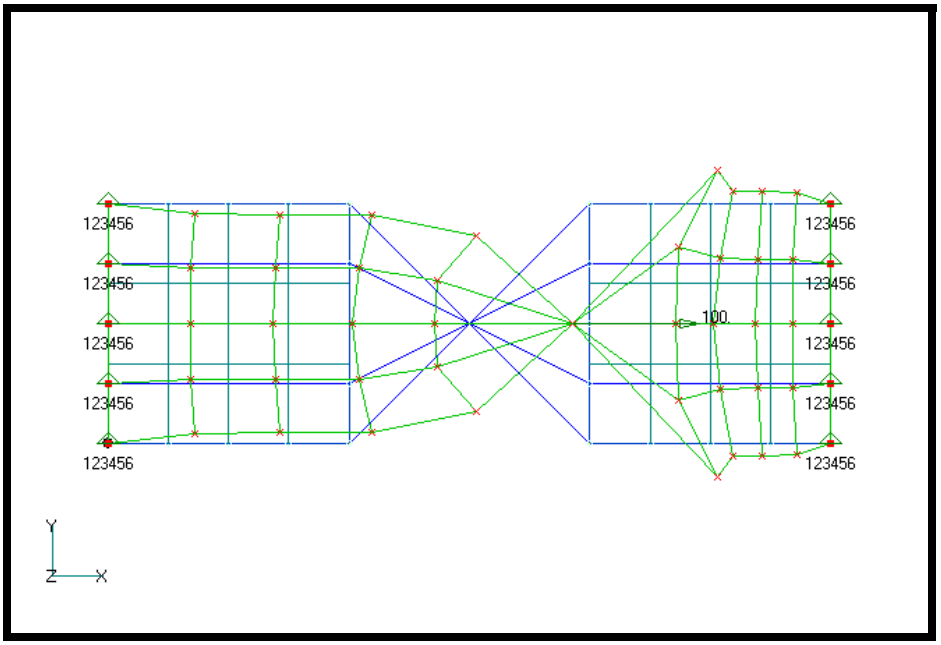
Deformation: *(pulldown)* **1..Total Translation**

OK

OK

Place the cursor on any of the nodes on the plate edges to see the displacement results. Now notice that the nodes on the edge of the plate do have y displacement with the RBE3 element, these nodes are free to move relative to each other.

Figure 34.3 - Deformed plot for Case 2.



View/Select... <F5>

Deformed Style: ● **Vector**

Deformed and Contour Data...

Deformation: **151..Total MultiPointForce**

OK

OK

The nodal results read from the *Entity Query* are as follows:

	MPC Forces (set to 151..Total MultiPoint Force)		**Displacement** (set to 1..Total Translation)	
Node	**T1**	**T2**	**X**	**Y**
9	**10**	**0**	**3.23e-5**	**-8.11e-6**
10	**10**	**0**	**2.23e-5**	**-4.14e-5**
11	**10**	**0**	**2.17e-5**	**0**
12	**10**	**0**	**2.23e-5**	**4.14e-6**
13	**10**	**0**	**3.23e-5**	**8.11e-6**
30	**10**	**0**	**3.23e-5**	**8.11e-6**
29	**10**	**0**	**2.23e-5**	**4.14e-5**
28	**10**	**0**	**2.17e-5**	**0**
27	**10**	**0**	**2.23e-5**	**-4.14e-6**
26	**10**	**0**	**3.23e-5**	**-8.11e-6**

Notice that the 100 lb force is evenly distributed to all the averaging nodes.

17. Now make the middle node dependent, while using three interpolation factors.

View/Select... <F5>

Deformed Style: ● **None**

OK

Delete/Model/Element...

Click on the element between the two plates.

OK

Yes

View/Autoscale <Ctrl+A>

Model/Element...

Interpolation: ☒ **Factor** **1**

DOF: ☒ **TX** ☒ **RX**
☒ **TY** ☒ **RY**
☒ **TZ** ☒ **RZ**

Reference/Node: **<select center node>**

DOF: ☒ **TX** ☒ **RX**
☒ **TY** ☒ **RY**
☒ **TZ** ☒ **RZ**

The following requires you to select specific nodes. The number of these nodes can be seen in **Figure 34.1**.

Nodes...

Entity Selection: **+9, +13, +26, +30**

OK

Interpolation: ☒ **Factor** **2**

Nodes...

Entity Selection: **+10, +12, +27, +29**

OK

Interpolation: ☒ **Factor** **3**

Nodes...

Entity Selection: **+11, +28**

OK

OK

Cancel

18. Repeat **Step 11** to analyze the model.

19. After the analysis, MSC.N4W automatically reads in the output results, which are available for a wide array of graphical and numerical postprocessing.

View/Select... <F5>

Deformed Style: ● **Deform**

Deformed and Contour Data...

Output Set: *(pulldown)* **3..MSC/NASTRAN Case 1**

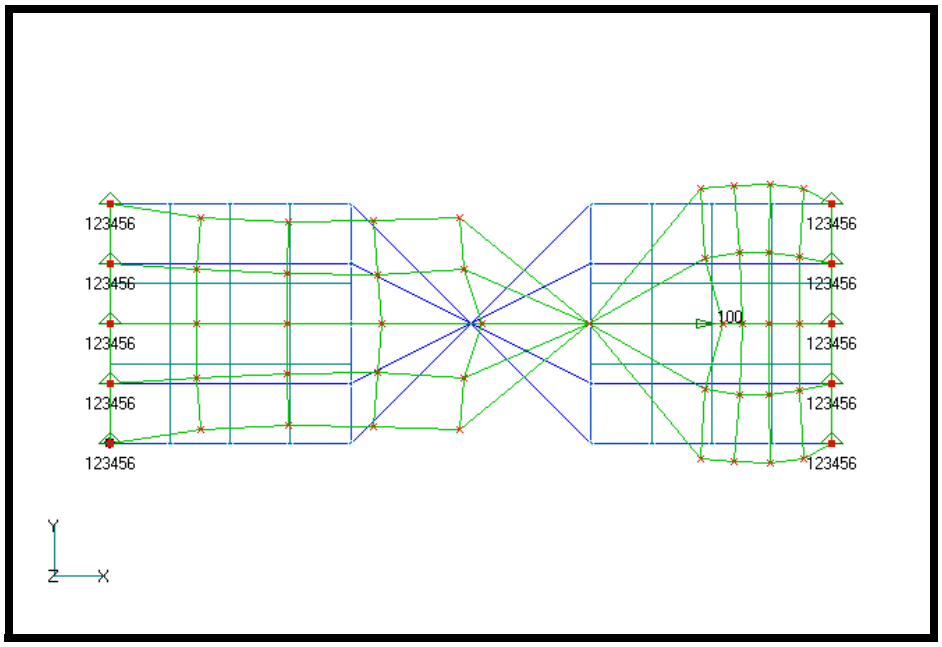
Deformation: *(pulldown)* **1..Total Translation**

OK

OK

Place the cursor on any of the nodes on the plate edges to see the displacement results.

Figure 34.4 - Deformed plot for Case 3.



Now to see the results of the multi-point forces, **right click** on the screen.

View/Select... <F5>

Deformed Style: ● **Vector**

Deformed and Contour Data...

Deformation: **151..Total MultiPointForce**

OK

OK

The nodal results read from the *Entity Query* are as follows:

	MPC Forces (set to 151..Total MultiPoint Force)		**Displacement** (set to 1..Total Translation)	
Node	**T1**	**T2**	**X**	**Y**
9	**5.55**	**0**	**2.31e-5**	**-2.95e-6**
10	**11.11**	**0**	**2.38e-5**	**-1.17e-6**
11	**16.67**	**0**	**2.78e-5**	**0**
12	**11.11**	**0**	**2.38e-5**	**1.17e-6**
13	**5.55**	**0**	**2.31e-5**	**2.95e-6**
30	**5.55**	**0**	**2.31e-5**	**2.95e-6**
29	**11.11**	**0**	**2.38e-5**	**1.17e-6**
28	**16.67**	**0**	**2.78e-5**	**0**
27	**11.11**	**0**	**2.38e-5**	**-1.17e-6**
26	**5.55**	**0**	**2.31e-5**	**-2.95e-6**

Notice how the interpolation factors have distributed the load.

In conclusion, RBE2 is a rigid element that references a group of nodes to an independent node and RBE3 is a rigid element that references a dependent node to a group of independent nodes.

This concludes this exercise.

File/Save

File/Exit

.