WORKSHOP 12

RBE2 vs. RBE3
Workshop Objectives

- Practice constructing RBE2 and RBE3 in Patran.
- Understand the difference between RBE2 and RBE3.
**Problem Description**

- A rectangular plate is fixed at its left and right edges. A rigid body element is in the middle of the plate. Model this two ways in one database using RBE2 and RBE3 elements.
- \( E = 10 \times 10^6 \) psi
- \( \nu = 0.33 \)
- \( t = 0.2 \) in
Suggested Exercise Steps

1. Create a new database and name it rbe2_vs_rbe3.db.
2. Create four rectangular surfaces.
3. Mesh the surfaces to create plate elements.
4. Create two additional nodes.
5. Create an RBE2 MPC.
6. Create an RBE3 MPC.
7. Apply Loads and Boundary Conditions.
8. Create material properties.
9. Create physical properties.
10. Run analysis with MSC.Nastran.
11. Read the results into MSC.Patran.
12. Plot the Von Mises stress and displacement.
13. Create vector marker plots to compare the RBE2 and RBE3 elements.
Step 1. Create New Database

Create a new database called rbe2_vs_rbe3.db

a. File / New.
b. Enter rbe2_vs_rbe3 as the file name.
c. Click OK.
d. Choose Default Tolerance.
e. Select MSC.Nastran as the Analysis Code.
f. Select Structural as the Analysis Type.
g. Click OK.
Step 2. Create Rectangular Surfaces

Create a surface

a. Geometry: Create / Surface / XYZ.

b. Enter <10 10 0> for the Vector Coordinates List.

c. Enter [0 0 0] [0 15 0] [20 0 0] [20 15 0] for the Origin Coordinates List.

d. Click Apply.
Step 3. Mesh the Surfaces

Create a surface mesh

- a. Elements: Create / Mesh / Surface.
- b. Select the surfaces.
- c. Enter 2.5 for the Global Edge Length.
- d. Click Apply.
Step 4. Create Two Nodes

Create two nodes

- b. Enter 1000 for the Node ID.
- c. Enter [15 5 0] for the Node Location List.
- d. Click Apply.
- e. Repeat the procedure with 2000 as the Node ID and [15 20 0] as the Node Location.
- f. Click on the Node Size Icon
Step 5. Create RBE2

Define dependent nodes.


b. Enter 1000 for the MPC ID.

c. Click on Define Terms.

d. Disable Auto Execute.

e. Select the nodes on the lower inner edges, as shown.

f. Select all six degrees of freedom in the DOFs list.

g. Click Apply.
Step 5. Create RBE2

Define independent nodes
   a. Select Node 1000 in the Node List box.
   b. Click Apply.
   c. Click Cancel.
   d. Click Apply in the Finite Elements form.
Step 6. Create RBE3

Define dependent nodes.

a. Elements: Create / MPC / RBE3.
b. Enter 2000 for the MPC ID.
c. Click on Define Terms.
e. Select all six degrees of freedom in the DOFs list.
f. Click Apply.
Define independent nodes

a. Click on Create Independent.

b. Select the nodes on the upper inner edges, as shown.

c. Select UX, UY, and UZ in the DOFs list.

d. Click Apply.

e. Click Cancel.
Step 6. Create RBE3

Finish creating RBE3

a. Elements: Create / MPC / RBE3.

b. Click Apply.
Step 7. Apply Loads and Boundary Conditions

Create a boundary condition


b. Enter fixed as the New Set Name.

c. Click Input Data.

d. Enter <0 0 0> for Translations and Rotations.

e. Click OK.
Step 7. Apply Loads and Boundary Conditions

Apply the boundary condition

a. Click Select Application Region.

b. For the Geometry Filter select Geometry.

c. Set the Selection Filter to Curve or Edge and select an outer edge, as shown.

d. Click Add.

e. Repeat the procedure until all four edges have been added.

f. Click OK.

g. Click Apply.
Step 7. Apply Loads and Boundary Conditions

Create a load

a. Loads/BCs: Create / Force / Nodal.

b. Enter **point_load** as the New Set Name.

c. Click **Input Data**.

d. Enter `<100 0 0>` for Force.

e. Click **OK**.
Step 7. Apply Loads and Boundary Conditions

Apply the load
a. Click Select Application Region.
b. For the Geometry Filter select FEM.
c. Select Nodes 1000 and 2000.
d. Click Add.
e. Click OK.
f. Click Apply.
Step 8. Create Material Properties

Create an isotropic material
b. Enter aluminum for the Material Name.
c. Click Input Properties.
d. Enter 10e6 for the Elastic Modulus.
e. Enter 0.33 for the Poisson Ratio.
f. Click OK.
g. Click Apply.
Step 9. Create Physical Properties

Create physical properties
a. Properties: Create / 2D / Shell.
b. Enter plate as the Property Set Name.
c. Click Input Properties.
d. Click on the Select Material Icon.
e. Select aluminum as the material.
f. Enter 0.2 for the Thickness.
g. Click OK.
Apply the physical properties

a. Click in the Select Members box.
b. Rectangular pick all surfaces as shown.
c. Click Add.
d. Click Apply.
Step 10. Run Linear Static Analysis

Analyze the model

b. Click Subcases.

c. Choose Default from Available Subcases.

d. Click Output Requests.

e. Select Multi Point Constant Forces.

f. Click OK.

g. Click Apply.

h. Click Cancel.

i. Click Apply.
Step 11. Read Results into MSC.Patran

Attach the results file

a. Analysis: Access Results / Attach XDB / Result Entities.
b. Click Select Results File.
c. Choose the results file rbe2_vs_rbe3.xdb.
d. Click OK.
e. Click Apply.
Step 12. Plot Stress and Displacement

Create a quick plot

a. Results: Create / Quick Plot.
b. Select Stress Tensor as the Fringe Result.
c. Select Displacements, Translational as the Deformation Result.
d. Click Apply.
Step 13. Plot Vector Markers

Create a marker plot

a. Click on **Reset Graphics**

b. Results: Create / Marker / Vector.

c. Select **MPC Constant Forces, Translational** as the Vector Result.

d. Select **Show As Resultant**.

e. Click **Apply**.
Step 13. Plot Vector Markers

Create a marker plot

a. Results: Create / Marker / Vector.

b. Click Reset Graphics.

c. Select MPC Constant Forces, Translational as the Vector Result.

d. Select Show As Component.

e. Select only \text{XX} as the Component.

f. Click Apply.
Step 13. Plot Vector Markers

Create a marker plot

a. Results: Create / Marker / Vector.

b. Select MPC Constant Forces, Translational as the Vector Result.

c. Select Show As Component.

d. Select YY as the component.

e. Click Apply.