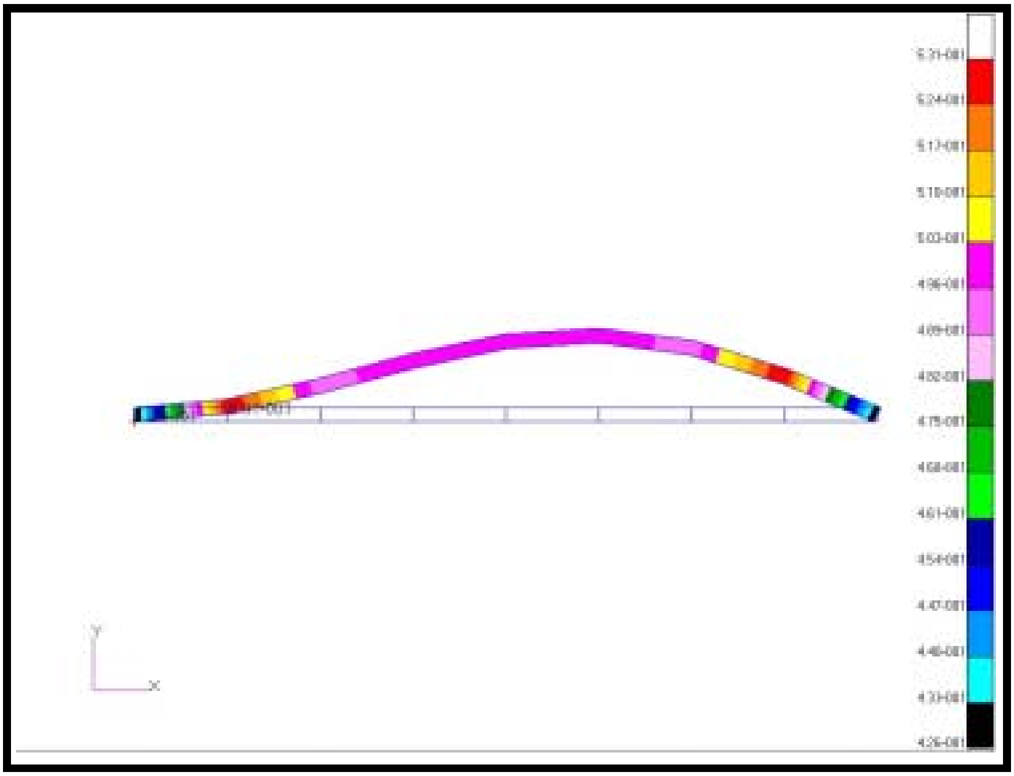
LESSON 17

Buckling of a Fixed Pinned Beam

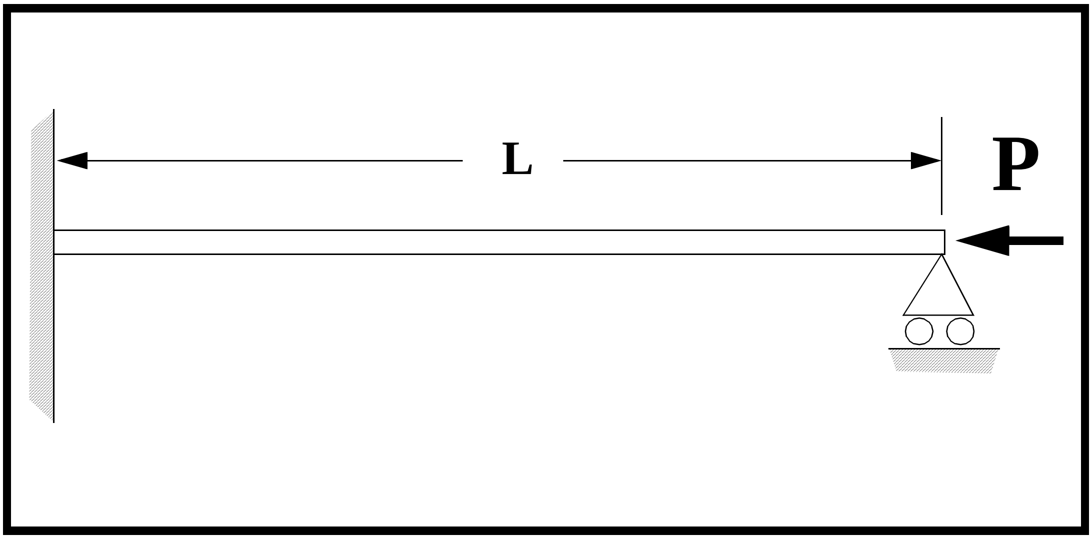


Objectives:

- Determine the eigenvalue buckling load and analyze in MSC.Marc.
- Compare the results to the theoretical prediction.

Model Description:

In this analysis you will be determining the eigenvalue buckling load for a fixed/simply - supported beam. After running the analysis, you will compare these results to the theoretical prediction. Once again, you will use the model built in *Exercise 1 - Build a Cantilever Beam* for this analysis.



Exercise Procedure:

1. Create a new database named **buckling_beam.db.**

File/New ...

New Database Name:	**buckling_beam.db**

OK

In the New Model Preference form set the *Analysis Code* to **MSC.Marc.**

Analysis Code:	**MSC.Marc**

OK

2. Import the old database. Use the cantilever beam model from the first part of this exercise.

File/Import ...

Object:	**Model**
Source:	**MSC.Patran DB**
Import File:	**cantilever_beam**

This will be the old database just created.

Apply

Close the summary form by selecting "OK."

OK

3. Now graphically display only the cantilever beam.

Group/Post...

Selected Groups to Post:	**cantilever_beam**

Apply

Cancel

4. Apply Additional Boundary Conditions

The left end of the beam is fixed in all active degrees of freedom. Now add a simple support to the right side of the beam so that it cannot deflect in the vertical direction as follows:

◼ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**fixed**

Input Data...

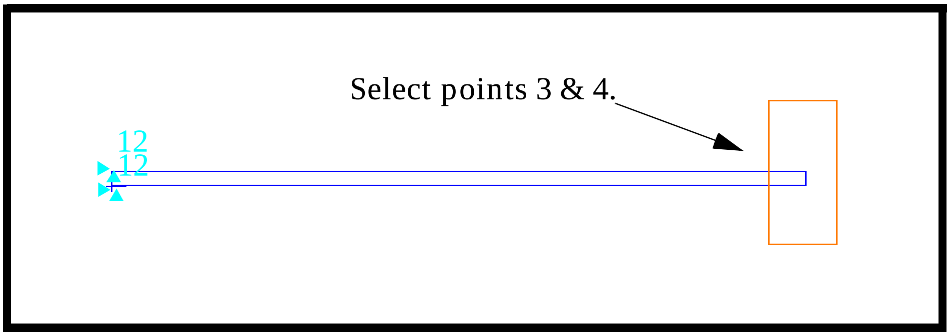
Translations:	**< , 0 >**

OK

Select Application Region...

Geometry Filter:	● **Geometry**
Select Geometric Entities:	**Point 3 4**

Figure 17.2 - Select points on the end for the simple support.



Add

OK

Apply

5. Add a unit compression Load.

Create a unit compression load that totals 1.0 lb at the beam’s end, and two 0.5 lb loads to the right side end nodes of the beam.

◼ **Load/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **compression_load**

Input Data...

Translations <T1 T2 T3>: **<-0.5, >**

OK

Select Application Region...

Geometry filter: ● **Geometry**

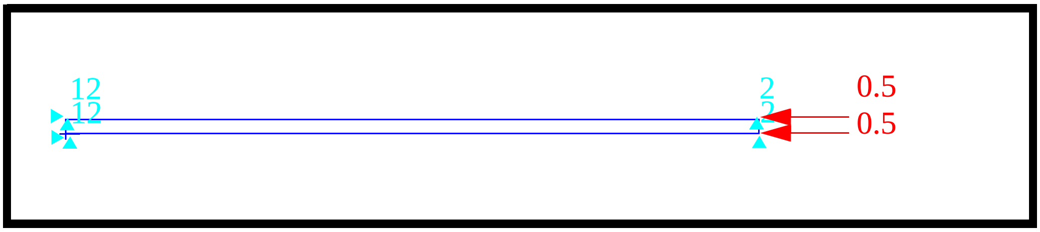
Select Geometric Entities: **point 3 4**

Add

OK

Apply

Figure 17.3 - Unit compression Load applied at right beam end.



6. Group Loads into Load Cases.

A number of loads and boundary conditions exist in the database now. We want to group these loads and boundary conditions into different sets called *load cases*. By doing this we can reference only the set of loads and boundary conditions

(LBCs) that are necessary for a particular analysis step. Open the **Load Cases** application and create two new load case for this buckling analysis.

■ **Load Cases**

Action:	**Create**
Load Case Name:	**Unit_Compression**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:	**Displ_fixed** **Displ_simply_supported** **Force_compression_load**

OK

Apply

Do it again for another load case with only the constraints.

Load Case Name:	**Buckling_Constraints**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:	**Displ_fixed** **Displ_simply_supported**

OK

Apply

7. Create static and buckling Analysis Load Steps.

To run the buckling analysis, we must create a two analysis **Load Steps**. The first step in a static analysis of the compression loading only. The second uses the results of the static analysis to determine the critical load and buckled deformation based on eigenvalue buckling analysis.

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**

Job Name:	**buckling_beam**
Load Step Creation...	
Job Step Name:	**Compression Step**
Solution Parameters...	
Linearity:	**Linear**
OK	
Select Load Cases...	
Available Load Cases:	**Unit_Compression**
OK	
Apply	

Repeat to setup the buckling step.

Job Step Name:	**Buckling Step**
Solution Type:	**Buckling**
Select Load Cases...	
Available Load Cases:	**Buckling_Constraints**
OK	
Apply	
Cancel	

Now select the steps in the *Analysis* form.

Load Step Selection...

Select the Jobs in step order. First select **Compression Step** and then **Buckling Step**. Unselect **Default Static Step** from the *Selected Job Steps* Form.

Existing Job Steps:	**Compression Step** **Buckling Step**
Selected Job Steps:	**Default Static Step**
OK	
Apply	

Again, you will need to monitor the analysis for job completion. After the job starts to run, MSC.Marc creates several files that can be used to monitor the job and verify that the analysis has run correctly. The ***buckling.log*** is an ASCII file which contains Element, Loads & Boundary Conditions, Material Translation, Step Control parameters, Equilibrium and Error information. When the job completes, this file contains an *Analysis Summary* which summarizes the error and iteration information. Another useful ASCII file is the ***buckling.sts*** file. This file contains a summary of job information; including step number, number of increments, number of iterations, total time of step, and time of a given increment. The ***buckling.out*** file contains a summary of any job errors. These files can be viewed during or after a job has completed. A more convenient method might be to use the **Analysis** application, **Monitor**.

Action: **Monitor**

Object: **Job**

View Status File...

After the job has finished, a successful completion will end with the line: Job ends with exit number: 3004

8. Read in the results when analysis job is finished.

■ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Attach**

Available Jobs: **buckling**

Select Results File... **buckling.t16**

OK

Apply

9. Post Process the results.

■ **Results**

Action: **Create**

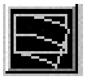
Object: **Quick Plot**

Selected Results Case: **... Mode=1, Fac=42907...**

Select Fringe Result: **Displacement, Translation**

Select Deformation Result: **Displacement, Translation**

Change the Display Properties for results. Click on the Deform Attributes icon.

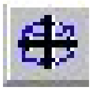


Scale Interpretation: **● Model Scale**

Scale Factor: **0.1**

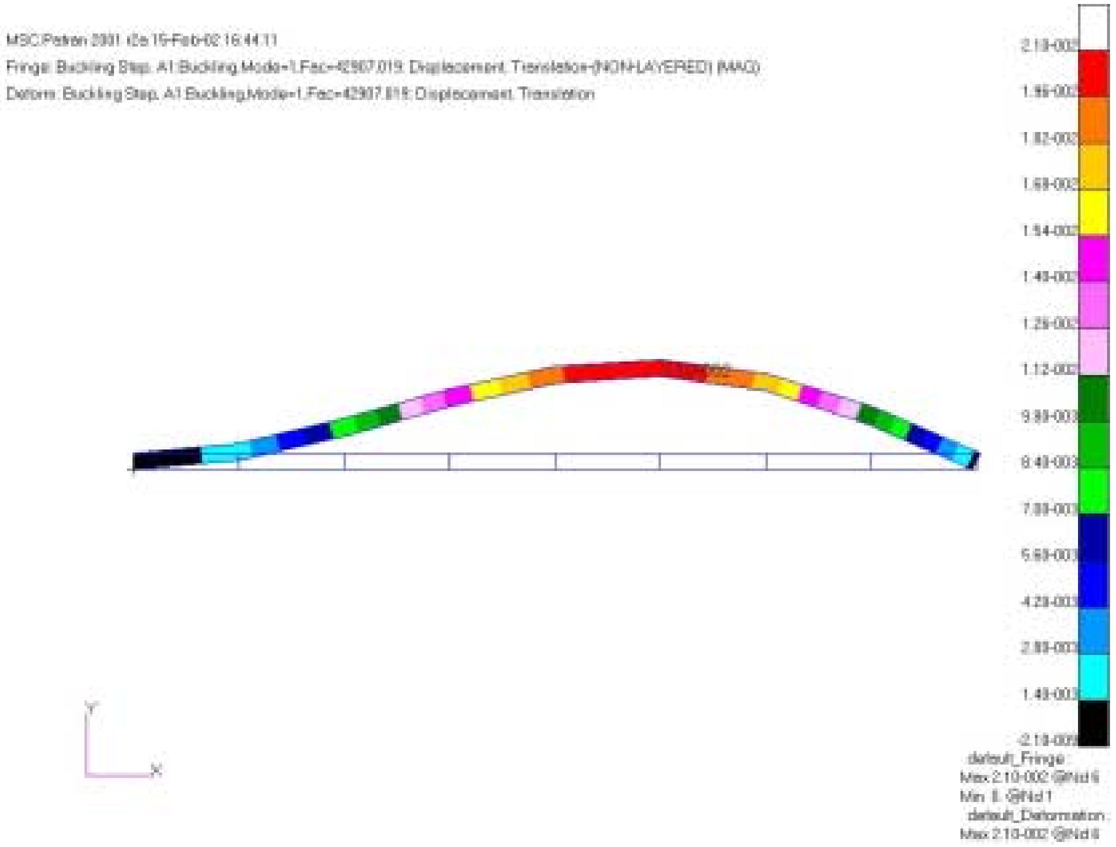
Apply

Choose the Fit View icon, or use the toolbar menu option under **Viewing/Fit View**.



The plot in Figure 17.4 should appear.

Figure 17.4 - Result of the Analysis



The total buckling load is the eigenvalue multiplied by the applied load. In this case, the total applied load is 1.0 and the eigenvalue can be found on the results case name on the results form.

$$P_{CR} = Eigen \times P_{Applied} =$$

The theoretical prediction for this case is

$$P_{CR} = \frac{\pi^2 EI}{L'^2}$$

C = A function of end constraint. For this case C = 2.05

$$L' = \frac{L}{\sqrt{C}} = 69.84$$

$$L' = \frac{L}{\sqrt{C}}$$

$$I = \frac{bh^3}{12} = \frac{(1)\cdot(2)^3}{12} = 0.6667 in^4$$

$$P_{CR} = \frac{\pi^2(3.0 \times 10^7)}{(69.84)^2} \times 0.6667 = 40470.84$$

Compare the results between the theoretical and finite element approach. The Eigenvalue is within six percent.

Theoretical	MSC.Marc
40471	42907

Close the database and quit PATRAN.

This concludes this exercise.