WS 03.3: Valve Body

Release 2022 R2

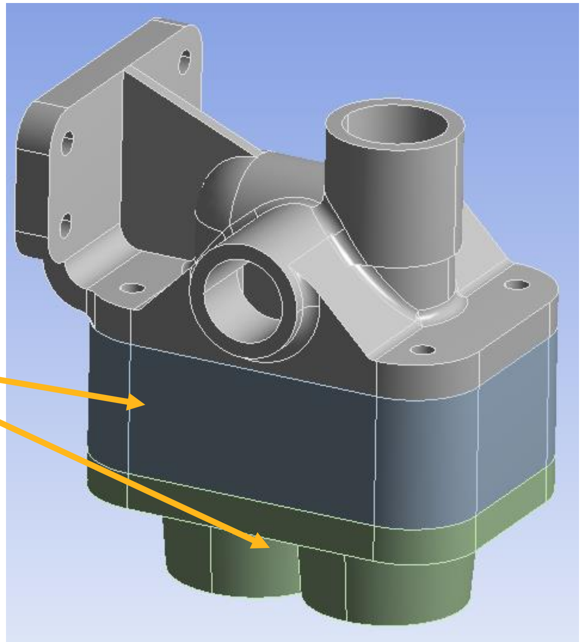
Please note:
- These training materials were developed and tested in Ansys Release 2022 R2. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.
- Although some workshop files may open successfully in previous releases, backward compatibility is somewhat unlikely and is not guaranteed.

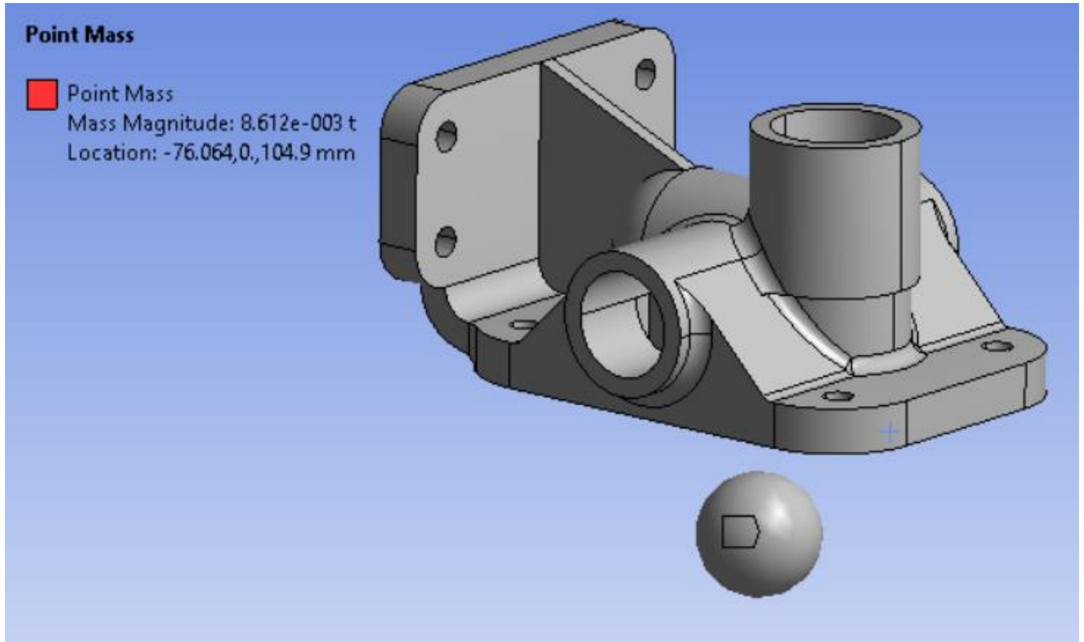
Ansys

Workshop 03.3 - Goals

- Upon completion of this workshop, you'll acquire an understanding of how to effectively replace components of an assembly with a point mass while maintaining the same modal characteristics of the assembly.
  - Simplify the model shown below by replacing two parts with a point mass while maintaining the same dynamic characteristics.
  - Learn how to effectively compare the two resulting models.

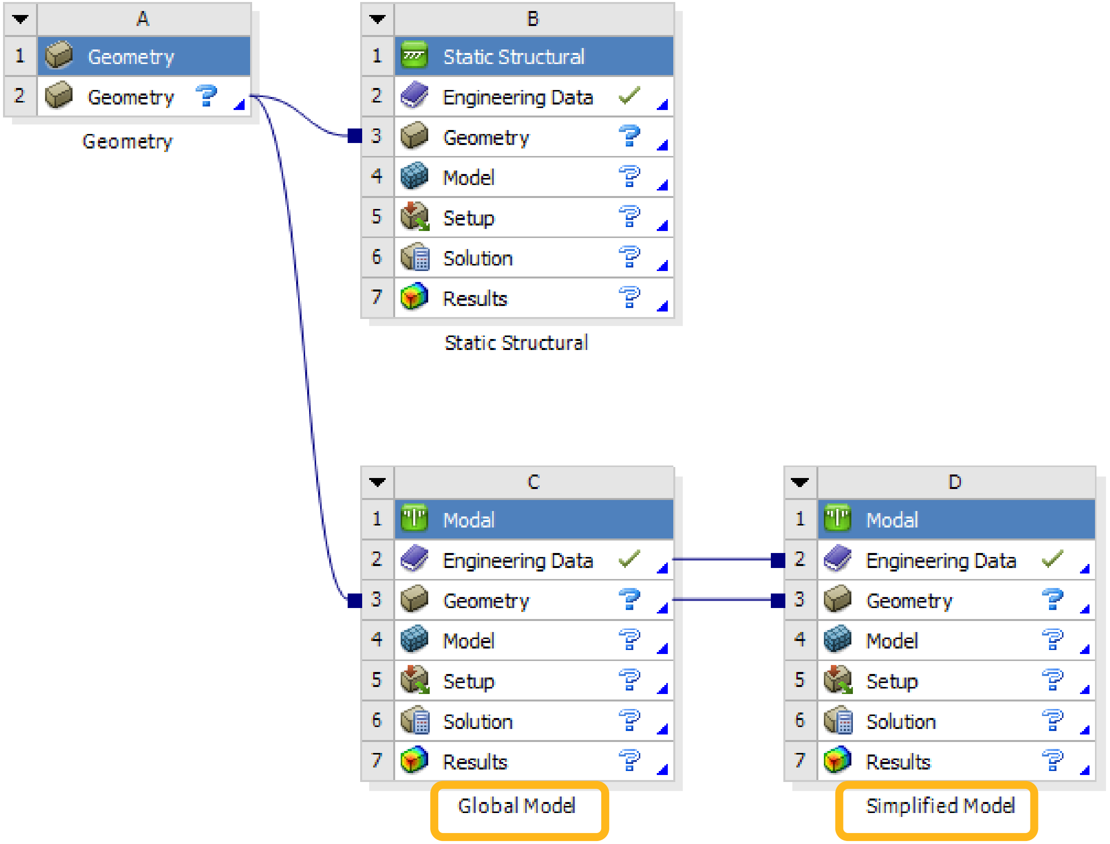
Replace these two components with an equivalent point mass





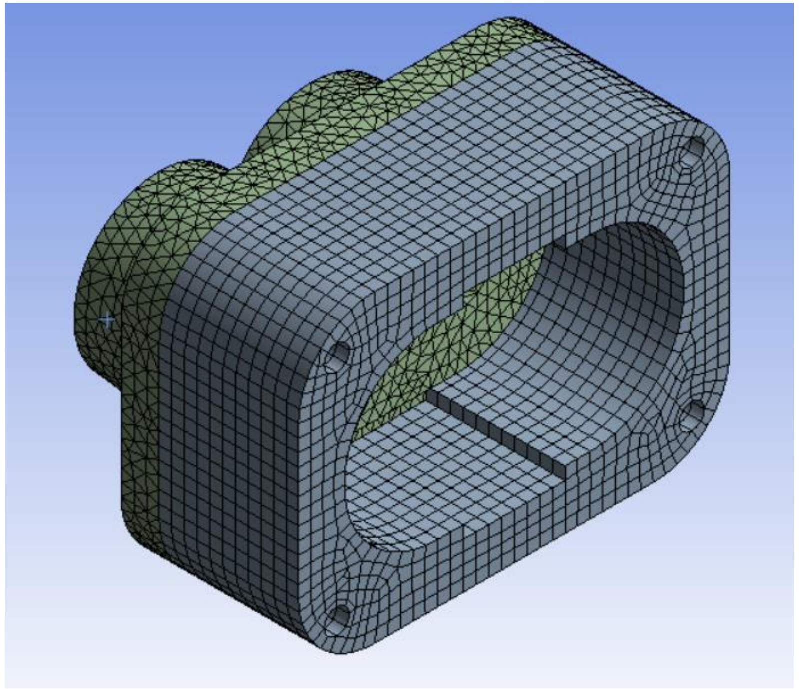
Workshop 03.3 - Project Schematic

- Begin a new Workbench session and, from the project schematic, insert component and analysis systems as shown here.
- Rename the two modal analysis systems as “Global Model” and “Simplified Model”.
- The Static Structural model will be used to compute the mass and moments of inertia of the components we will be replacing.



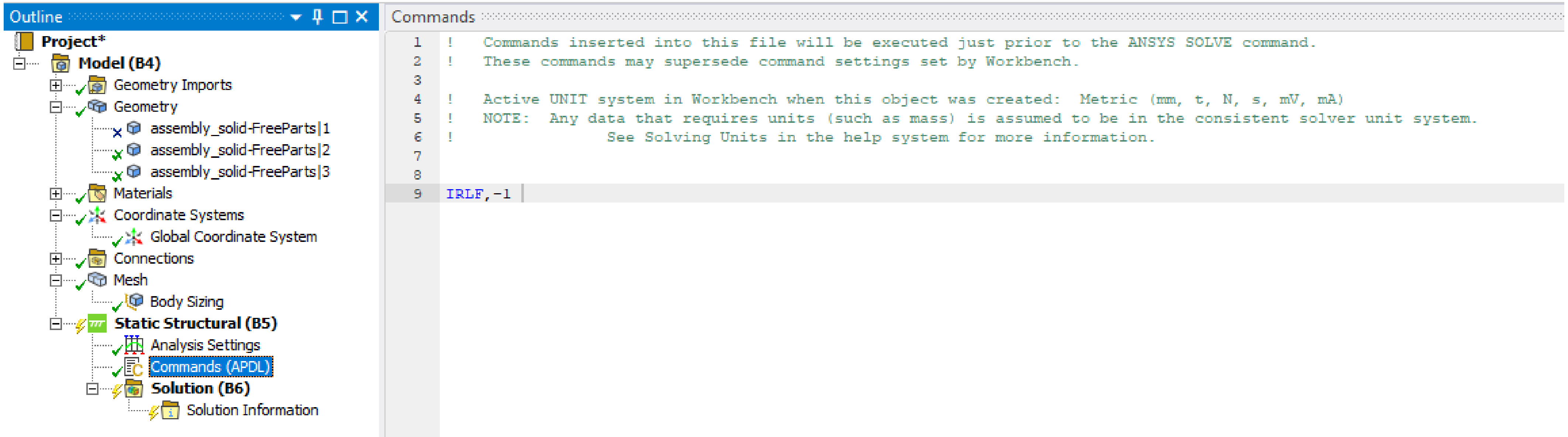
Workshop 03.3 – Static Structural

- Import the Geometry file "**assembly_solid.stp**"
- Launch Mechanical from the Static Structural system
  - Units: Metric (mm, t, N, s, mV, mA)
- Suppress the first solid, leaving the remaining two bodies for mass and inertia calculations.
- Insert a Mesh Body Sizing of 5 mm on the two remaining bodies and mesh the model.



Workshop 03.3 – Inertia Calculations

- From the Static Structural branch, insert a Commands object
  - Type the APDL Command "IRLF,-1" in the Commands window
  - The IRLF command will allow calculation of an accurate mass summary of the model without performing any other calculations, since the model contains no loads or constraints
- Solve the model



Note: warning messages will be produced from the solution, indicating lack of sufficient constraints generating solver pivot warnings. This is fine since we're just calculating a mass summary.

Workshop 03.3 – Inertia Calculations

Note: your result magnitudes may vary slightly throughout this workshop due to mesh and software release differences

- In Solution Information, search for and review the Precise Mass Summary

Worksheet

```
           ***********  PRECISE MASS SUMMARY  ***********

 TOTAL RIGID BODY MASS MATRIX ABOUT ORIGIN
             Translational mass                   Coupled translational/rotational mass
    0.86120E-02    0.0000         0.0000          0.0000        0.90341      -0.20289E-07
     0.0000       0.86120E-02     0.0000         -0.90341       0.0000       -0.65507
     0.0000        0.0000        0.86120E-02      0.20289E-07   0.65507        0.0000
                                                         Rotational mass (inertia)
                                                   127.18       0.44686E-06    68.685
                                                  0.44686E-06    173.36      -0.47303E-05
                                                   68.685     -0.47303E-05    67.981

 TOTAL MASS = 0.86120E-02
   The mass principal axes coincide with the global Cartesian axes

 CENTER OF MASS (X,Y,Z)=   -76.064        0.23559E-05    104.90

 TOTAL INERTIA ABOUT CENTER OF MASS
       32.412      -0.10964E-05  -0.32920E-01
    -0.10964E-05    28.761       -0.26019E-05
    -0.32920E-01  -0.26019E-05    18.153
   The inertia principal axes coincide with the global Cartesian axes
```

: mass of the model

: inertia in the global coordinate system (impossible to use it in Mechanical because no possibility to enter extra diagonal term)

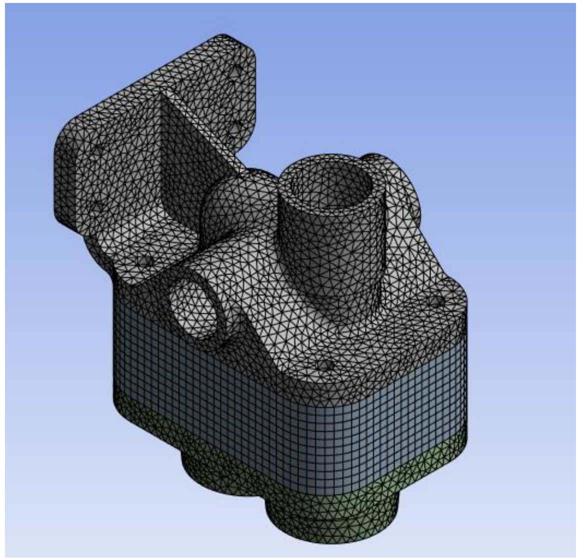
: center of mass (use in order to put the right center of gravity in the simplified model)

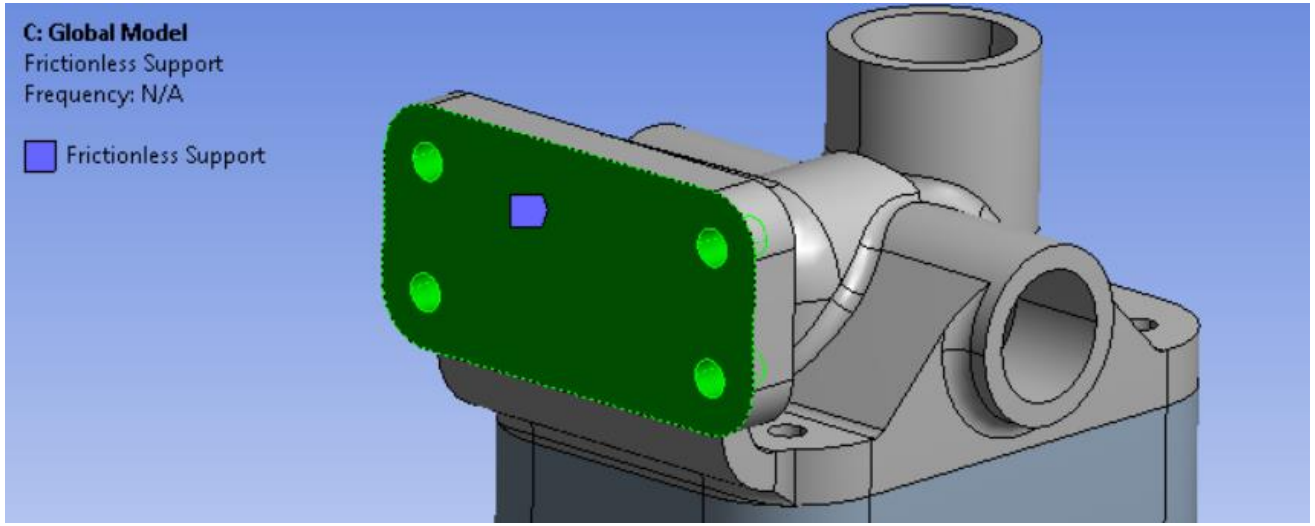
: inertia in the principal axes and orientation vector for the principal axes

Ansys

Workshop 03.3 - Global Model

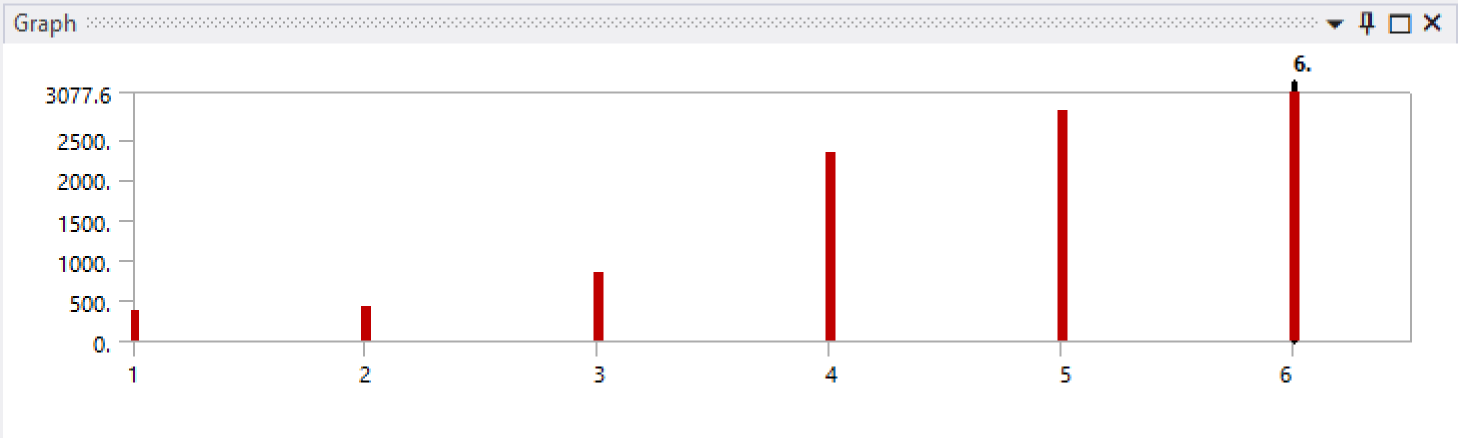
- We'll use this model as a baseline with which to compare our Simplified Model. We'll run a modal analysis and calculate the first 6 modes.
- Launch Mechanical from the Global Model analysis system and set the units system to "Metric (mm, t, N, s, mV, mA)".
- In the Mesh branch, insert a Body Sizing of 5 mm for the 3 bodies.
- Apply a frictionless support to the 9 faces shown below and solve.





Workshop 03.3 - Global Model

- Review the natural frequencies and the mode shapes

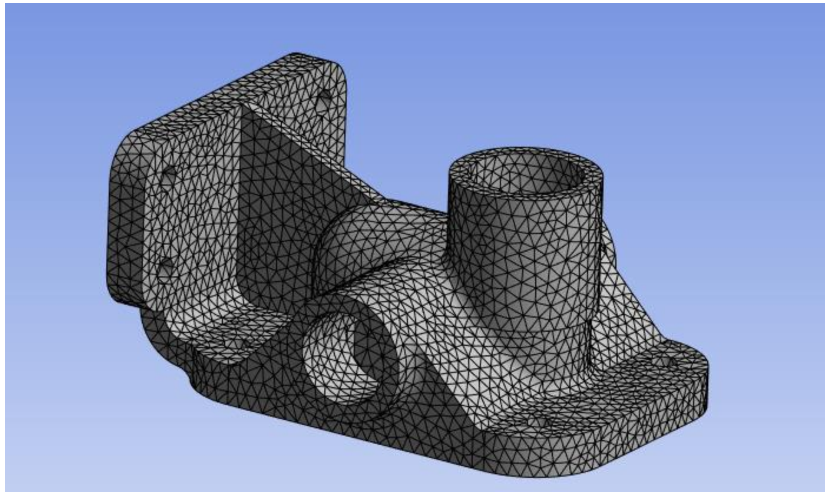


Tabular Data

	Mode	Frequency [Hz]
1	1.	380.73
2	2.	426.66
3	3.	834.03
4	4.	2332.7
5	5.	2845.5
6	6.	3077.6

Workshop 03.3 - Simplified Model

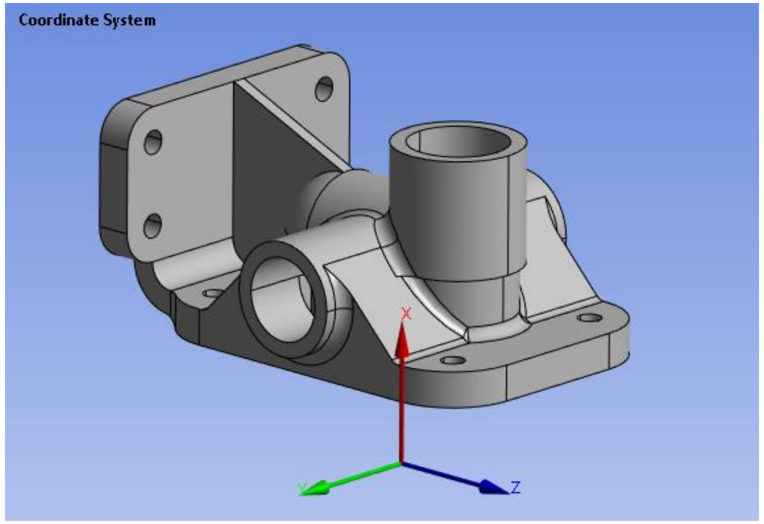
- We'll use this model to replace two of the components of the assembly with a point mass.
  - The mass and inertia properties obtained from the Static Structural analysis will be used in the definition of the point mass.
- Launch Mechanical from the Simplified Model analysis system and set the units system to “Metric (mm, t, N, s, mV, mA)”
- Suppress the second and the third bodies and apply the same 5mm Body Sizing mesh control to the remaining unsuppressed body in the model.



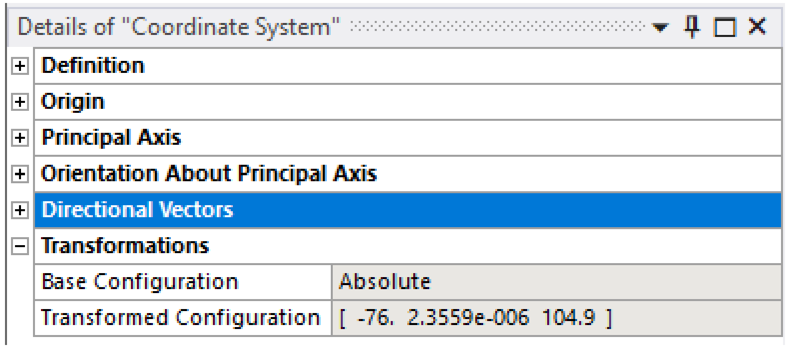
Workshop 03.3 - Simplified Model

- Create a new cartesian coordinate system at the center of mass obtained from the Static Structural analysis.
  - Global Coordinates:
    - Origin X: -76.064 mm
    - Origin Y: .23559e-5 mm
    - Origin Z: 104.9 mm

center of mass from slide 6

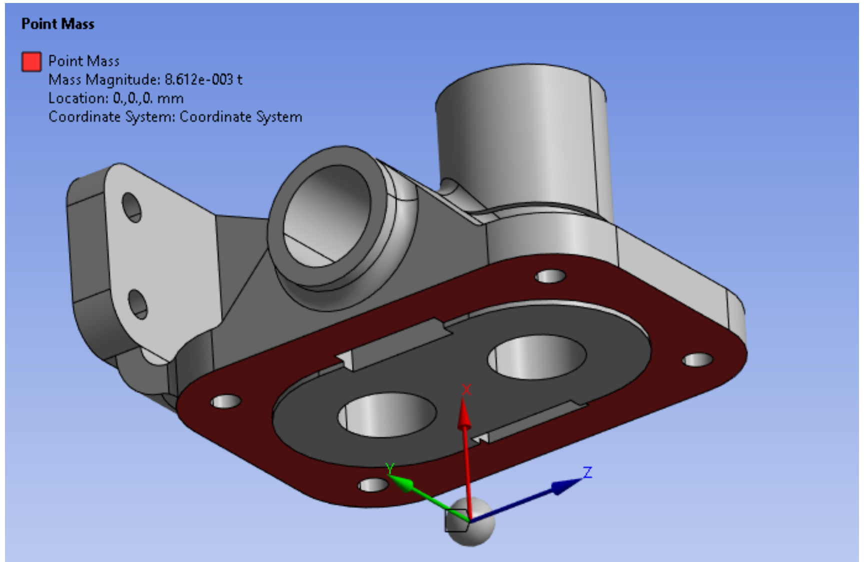


Note: In this example, the inertia principal axes coincide with the global cartesian axes (check slide 6). In other cases, the angles of the principal axes must be hand-calculated and defined as Transformations (Rotate Z, Rotate Y, Rotate X) of the Coordinate System.



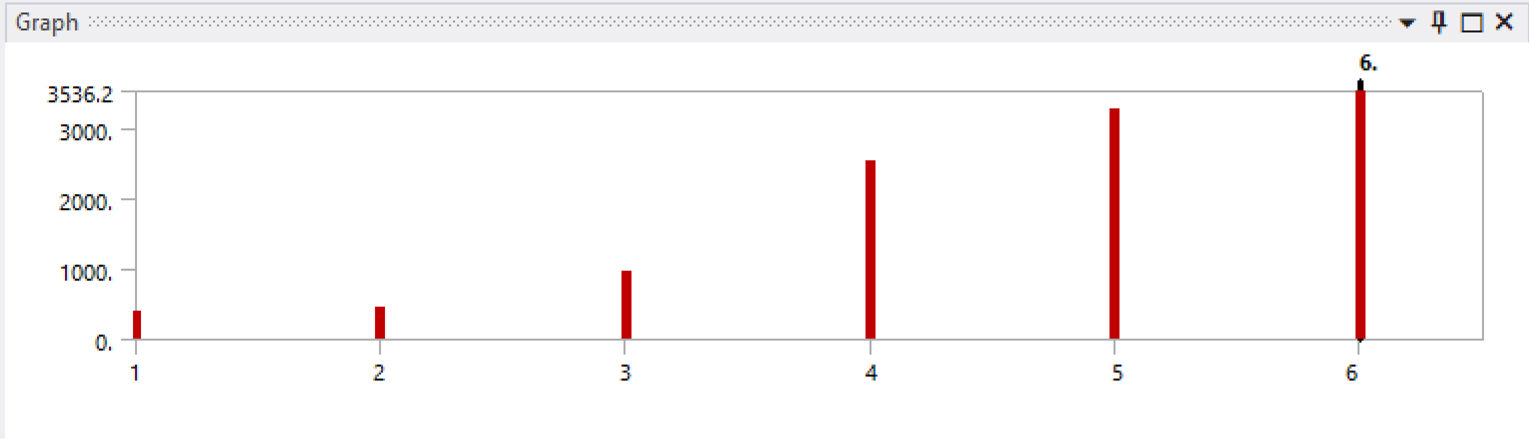
Workshop 03.3 - Simplified Model

- Insert a point mass to replace the two bodies and attach it to the surface shown.
- Reference the origin (0,0,0) of the newly created coordinate system from the previous step.
- Mass and Inertia properties obtained from Static Structural analysis
  - Mass: 8.612e-3 t
  - Mass Moment of Inertia X: 32.412 t-mm^2
  - Mass Moment of Inertia Y: 28.761 t-mm^2
  - Mass Moment of Inertia Z: 18.153 t-mm^2
  - Behavior: Rigid (since the bodies were connected by bonded contact)



Workshop 03.3 - Simplified Model

- Apply the same frictionless support (see slide 7), Save the project as WS03.3, solve, and review results:

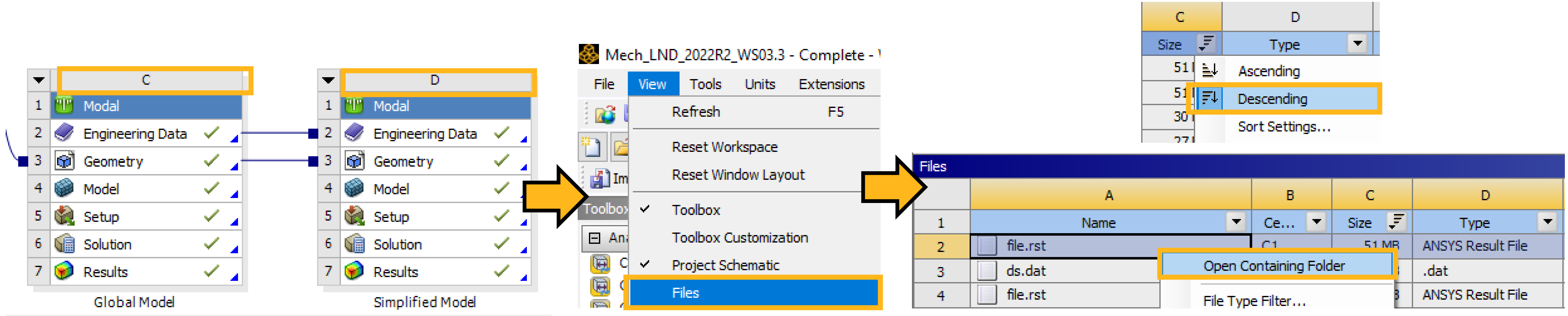


Tabular Data

	Mode	Frequency [Hz]
1	1.	386.17
2	2.	442.94
3	3.	965.05
4	4.	2537.6
5	5.	3287.3
6	6.	3536.2

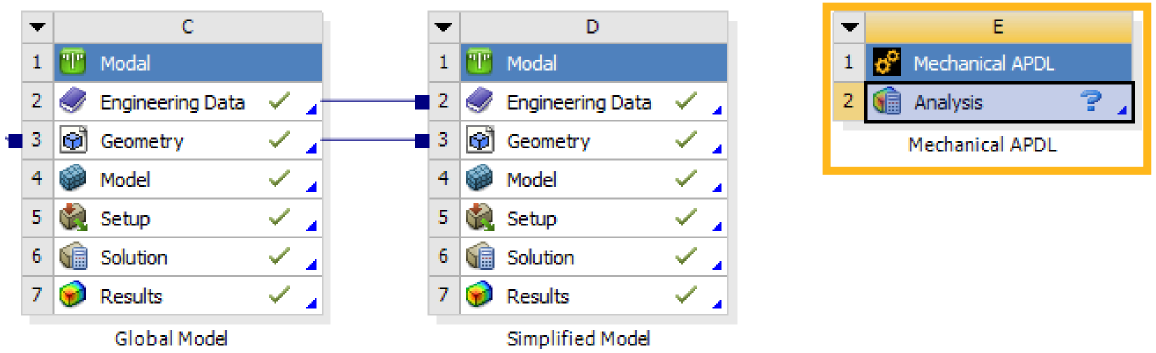
Workshop 03.3 - Comparison

- To compare the 2 models, we will use the APDL command RSTMAC to see the differences in the mode shapes (eigenvectors)
- Copy and paste files "file.rst" from the Global and Simplified Models to the same folder (rename to "file1.rst" for the Global Model and "file2.rst" for the Simplified Model).
  - To locate these files, return to Project Schematic, enable visibility of files (View menu > Files), then sort files by descending size.
  - Right mouse button on each file in the list, and choose "Open Containing Folder"; copy the file…
  - … and paste each file into the "user_files" folder associated with this project (located 3 directories above the current), renaming as above.

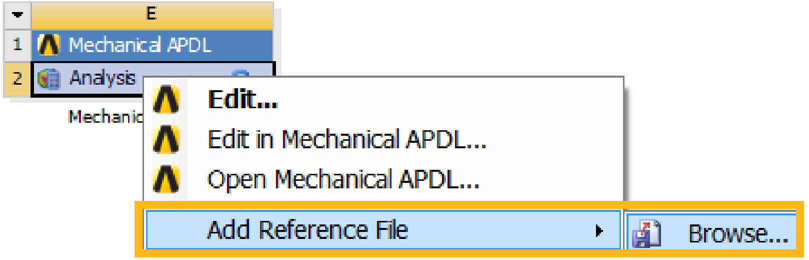


Workshop 03.3 - Comparison

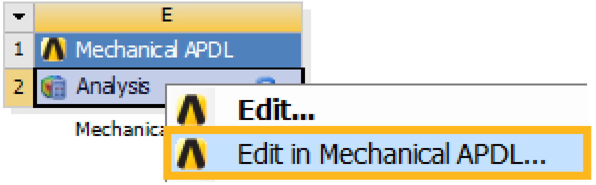
- From the Project, Insert a *Component System > Mechanical APDL* to the right of the Simplified Model Schematic



- Right mouse button on the Analysis cell, Add Reference File, and choose file1.rst from the user_files directory. Repeat again for file2.rst.

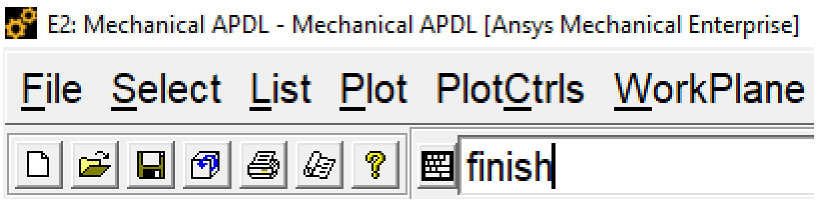


- Right mouse button on the Analysis cell, Edit in Mechanical APDL…



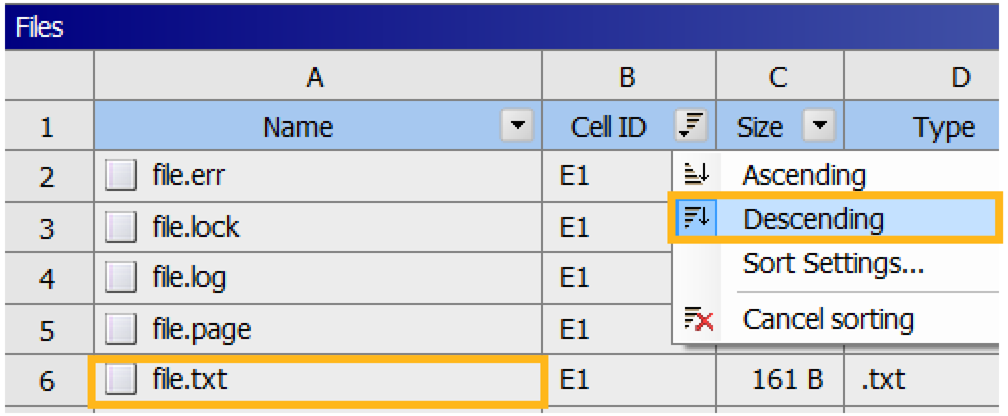
Workshop 03.3 - Comparison

- From within the Mechanical APDL command prompt, type the following commands that will create a “file.txt” summarizing the results of the comparison:




```
finish
/clear
/output,file,txt
/post1
rstmac,file1,,,file2,,,,,,1
/output
```

- From the Project Schematic, locate the “file.txt” by sorting the file listing by descending Cell ID:



Workshop 03.3 - Comparison

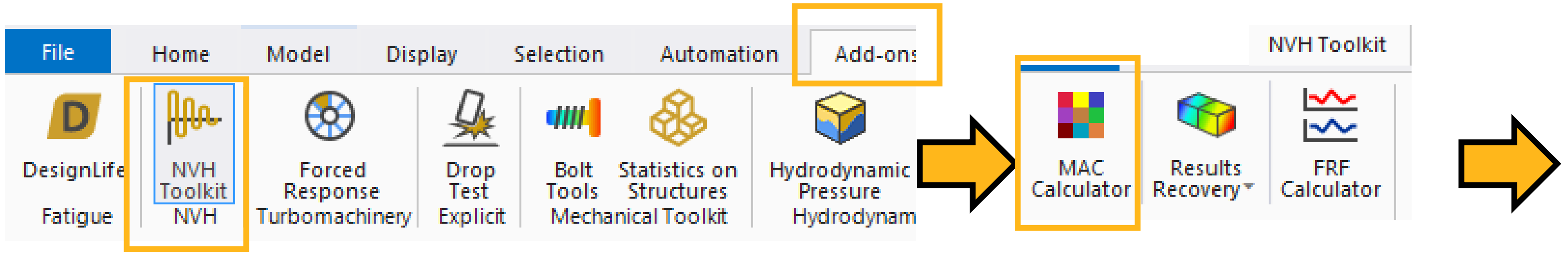
- Open file “file.txt” (created via the APDL command /OUTPUT):

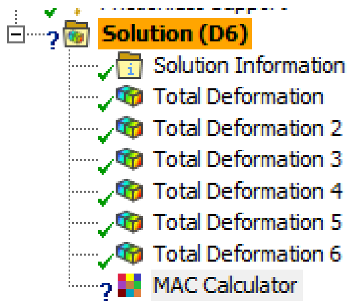
```
 READING FILE file1.rst
  Number of nodes                           =   169996
  Number of elements                        =   105170
  Number of degrees of freedom per node =        3
  Number of solution sets                   =        6
  Complex solutions                         =      no


 READING FILE file2.rst
  Number of nodes                           =    85892
  Number of elements                        =    56305
  Number of degrees of freedom per node =        6
  Number of solution sets                   =        6
  Complex solutions                         =      no

*** NOTE ***                            CP =       1.328   TIME= 14:46:31
Nodes matching in RSTMAC command succeeded.
 85890 pairs of nodes are within the tolerance (.01).

  ****** Modal Assurance Criterion (MAC) VALUES ******
                 Solutions are real
   Rows:       6 substeps in load step  1 from FILE1
   Columns:    6 substeps in load step  1 from FILE2

            1       2       3       4       5       6
    1   1.000   0.000   0.000   0.032   0.166   0.000
    2   0.000   1.000   0.378   0.000   0.000   0.172
    3   0.000   0.395   0.997   0.000   0.000   0.185
    4   0.082   0.000   0.000   0.980   0.074   0.000
    5   0.129   0.000   0.000   0.341   0.938   0.000
    6   0.000   0.196   0.175   0.000   0.000   0.988

*** NOTE ***                            CP =       6.922   TIME= 14:46:37
Solutions matching in RSTMAC command succeeded.
 6 pairs of solutions have a Modal Assurance Criterion (MAC) value
greater than the smallest acceptable value (.9).

  ********************************** MATCHED SOLUTIONS **********************************
  Substep in           Substep in           MAC value     Frequency         Frequency
  FILE1                FILE2                              difference (Hz)   error (%)
        1                    1                1.000       -0.54E+01          1.4
        2                    2                1.000       -0.16E+02          3.8
        3                    3                0.997       -0.13E+03         15.7
        4                    4                0.980       -0.20E+03          8.8
        5                    5                0.938       -0.44E+03         15.5
        6                    6                0.988       -0.46E+03         14.9
```

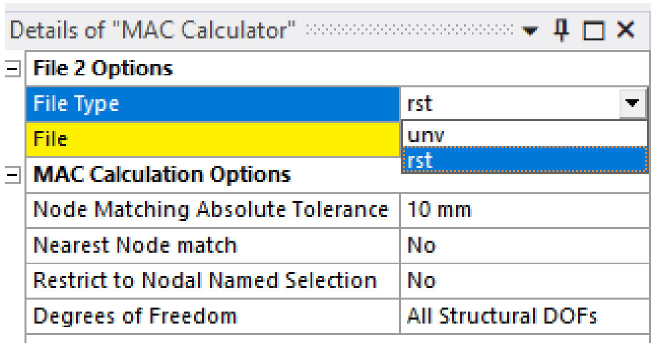
The matrix shown here is the scalar product of the corresponding eigenvectors of each model. If the two models have exactly the same eigenvectors the diagonal term will be 1.000; if the two eigenvectors are completely different (that is, uncorrelated) the diagonal term will be 0.000.

Workshop 03.3 – Comparison (MAC Calculator)

- The MAC Table can also be generated using the MAC Calculator* in the NVH Toolkit Addon.
- Activate the NVH Toolkit from Add-ons Tab. Select the MAC calculator to add it in the Solution object.





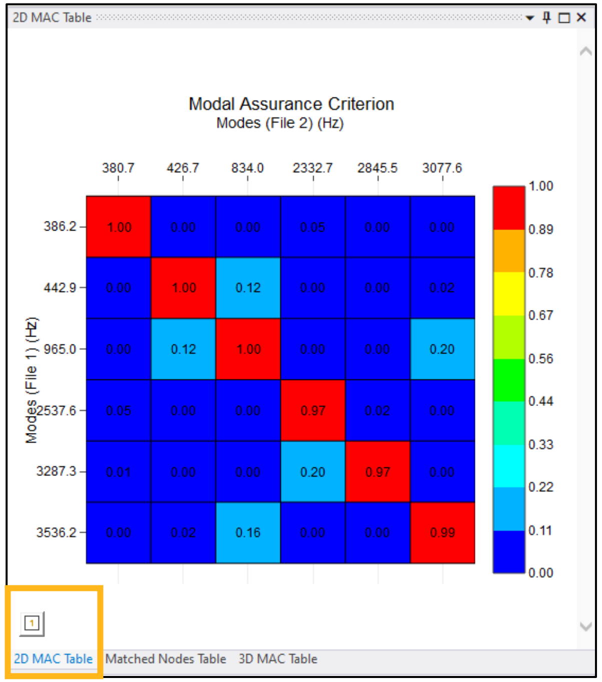


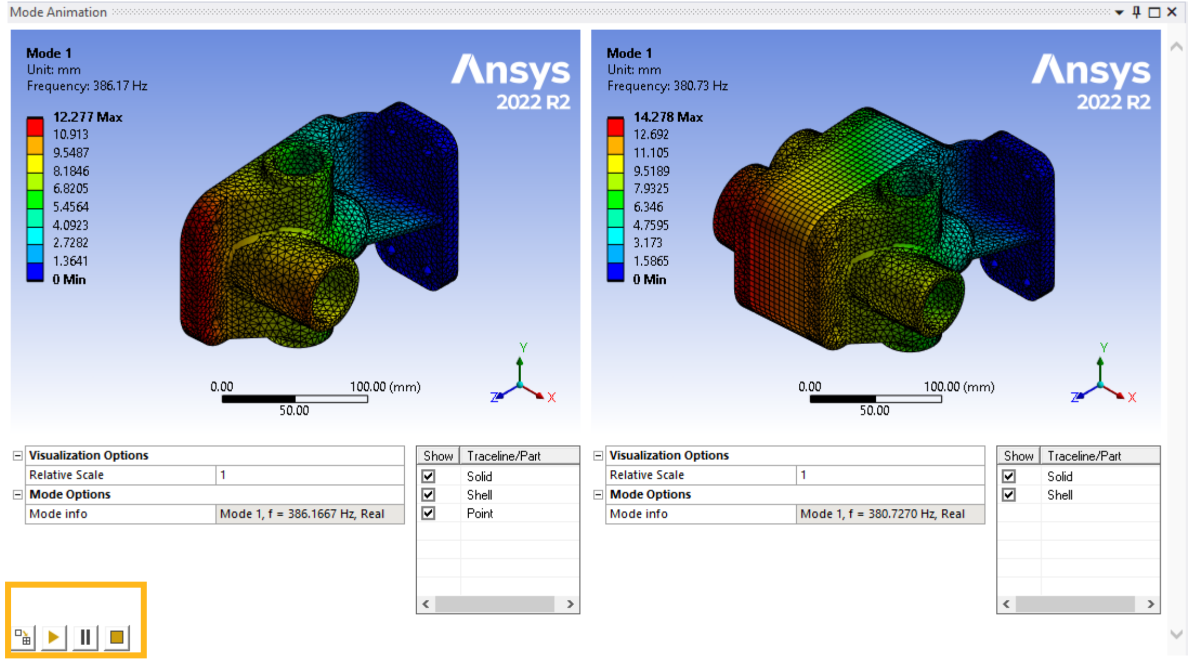
- Set the file type to rst, select the file.rst file for the SYS 1 from the path ...M03 - Modal\Mech_LND_2022R2_WS03.3 \dp0\SYS-1\MECH\file.rst and Right click to generate the solution.

**Note this feature is only available in Ansys Mechanical 2022R2 versions and later*

Workshop 03.3 – Comparison (MAC Calculator)

- Select the 2D MAC Table and click on the show numbers button ( ) to display the values.
- The 2D MAC Table shown is the graphical representation of the matrix obtained using the APDL commands output.
- Click on the table values to generate and animate modal comparison of the two models.





End of presentation