

Introduction to ANSYS Mechanical

Realize Your Product Promise®



Our goal is to investigate the vibration characteristics of the machine frame shown here. We want to solve 2 modal analyses using different mounting points on the frame.



ANSYS Assumptions

The frame contains 8, 20 mm diameter mounting holes. In the first analysis the frame will be constrained by all 8 holes. In the second analysis we'll only constrain the corner holes. To simplify the setup we'll create several named selections.



ANSYS Project Schematic

- 1. From the Toolbox double click "Modal" to create a new system.
- 2. RMB the "Geometry" cell and browse to "Machine_Frame.stp".
- 3. Double click "Model" to open the Mechanical application.









- 4. Set the working Unit System:
 - Units > Metric (mm, kg, N, s, mV, mA)



Mesh control

- 5. Creating the named selections:
 - a. Select one of the small edges than RMB > create named selection
 - b. In the Named Selection details choose Apply geometry items of same size
 - c. Repeat a. and b. with one of the small faces and rename the named selection: 'faces'

	5a.	5b.	Selection Name	50	С.
Insert Gr. To Gr. To Gr. To Hide Body (79) Hide All Other Bodies Hide Tree Based On Visible Bodies Gr. Suppress Body Suppress All Other Bodies Suppress All Other Bodies Suppr			Apply selected geometry Apply geometry items of same: Size Type Location X Location Y Location Z OK Cancel	G TO G TO G Generate Named Selection Filde Body (F9) Filde The Bod ON Visible Bodies G Suppress Body Suppress All Other Bodies Filde Tree (F8) Filde Tree (F8) Formatic View Set Restore Cellault Cursor Mode View Carsae Coordinate System Carsae Carsae Selection G Select All (Ctrl + A)	

ANSYS ... Mesh control

- 6. Changing the mesh controls:
 - a. Under mesh details RMB > insert > sizing and scope to the Named selection edges.
 - b. In the details of Sizing, change the type to Number of divisions and set this number to 1.





- 7. ...Changing the mesh controls:
 - a. Insert a face meshing and scope it to the named selection 'faces' and change the Internal Number of divisions to 1.
 - b. In the global sizing control of the mesh details set the Element Size to 30 mm.



7b.	□ Mesh Edge Sizing Face Meshing ▼						
D							
] Display						
	Display Style	Body Color					
-	Defaults						
	Physics Preference	Mechanical					
	Relevance	0					
-	Sizing						
	Use Advanced Size Function	Off					
_	Relevance Center	Coarse					
	Element Size	10.0 mm					
	Initial Size Seed	Active Assemb	ly				
	Smoothing	Medium					
	Transition	Fast					
	Span Angle Center	Coarse					
	Minimum Edge Length	5.0 mm					

Note : At this step, mesh is not as fine as required for analysis. Nevertheless, we will use this one for time reasons

© 2015 ANSYS, Inc. February 27, 2015

8

ANSYS Boundary conditions

- 8. Create a first Named Selection for the holes surfaces:
 - a. Highlight the Model branch, RMB > Insert > Named Selection.



b. In the Named Selection details change the scoping method to "Worksheet".



ANSYS ... Boundary conditions

- 9. Create the first Named Selection:
 - a. In the Worksheet, RMB > Add Row.
 - b. Pick the action "Add" and configure as shown below.
 - c. Generate.
 - d. Rename to "Eight Holes".

Knowing the holes are 20 mm diameter we use the "radius" criteria to create a named selection containing all of the mounting holes.

9d.

Generate			rate	9c.			
		Action	Entity Type	Criterion	Operator	Units	Value
9b.		Add	Face	Radius	Equal	mm	10
10 © 2	015 ANSY	Sinc Febr	uary 27 2015				



- **10. Create another Named Selection:**
 - a. Highlight the named selection "Eight Holes", RMB > Duplicate.
 - b. Rename the new named selection "Four Holes".
 - c. In the details verify the scoping method is still set to "Worksheet".









- **11. Create a second Named Selection:**
 - a. RMB and "Add Row".
 - b. Pick the action "Remove" and configure as shown below.

	Ge	enerate
	Action	Entity Type
	Add	Face
- 11a.		Add Row
	141	Insert

End	Holes		_						
	Gener	rate	11c.						
	Action	Entity Type	Criterion	Operator	Units	Value	Lower Bound	Upper Bound	Coordinate
	Add	Face	Radius	Equal	mm	10	N/A	N/A	N/A
	Remove	Face	Location X	Range	mm	N/A	300.	1200.	Global Coor
	11b.			1		1			

Using the "remove" operation and a range in the X direction we have eliminated the interior holes from the named selection.

... Boundary conditions

- 12. Apply supports to model (highlight the "Modal" branch (A5):
 - a. From the Supports menu select a "Cylindrical Support".

- **b.** Switch the scoping method to "Named Selection".
- **C.** From the named selection list choose "Eight Holes".
- d. Change the Tangential setting to "Free".



NNSYS



Set Options for Modal Analysis:

13. Highlight "Analysis Settings" to set the "Max Modes to Find" (defaults to 6 modes).

As a final check verify the status symbols next to the branches. All branches should have either:

- Yellow Lightening bolt (ready to be solved).
- Green check mark (fully defined).

DO NOT SOLVE YET!



ANSYS Project Schematic

- 14. Add a second modal analysis:
 - Return to the Workbench Project, highlight the "Setup" cell, RMB > Duplicate.





The schematic should look like this after duplicating.

ANSYS Environment

- **15. Change the supports in the second environment:**
 - a. Highlight the "cylindrical support" in the second environment (Modal (B5)).



Scoping Method Named Selection Named Selection Four Holes

Fixed

Fixed

Free

No

Cylindrical Support

15b.

Scope

Type Radial

Axial

Definition

Tangential

Suppressed

b. In the details pick the named selection "Four Holes" from the list.

16

ANSYS Environment

- **16. Solve both environments:**
 - a. Highlight the Model branch.
 - b. RMB > Solve.

Note, by solving from the Model branch, both environments will be solved.





- **17.** Select Mode shapes to view (repeat this for both Solution branches):
 - a. Click on the "Solution" branch. This will display the "Graph" and the "Tabular Data" showing a summary of the frequencies at which the modes occur.
 - **b.** In the "Graph" RMB > "Select All" to select all modes.
 - Note : This can be done from the "Tabular Data" as well.
 - **c.** RMB > "Create Mode Shape Results".





Comparing the 2 solutions shows very little difference in the 2 environments until the 6th mode.





Comparing the 6th mode from each solution shows the difference.

• Note: using the viewports "Horizontal Viewports" option you can display both results as shown here.



-
One Viewport
Horizontal Viewports
Vertical Viewports
Four Viewports



18. Create a histogram to compare the two results:

- a. Hold the CTRL key and highlight both Solution branches.
- b. Click the "Chart/Table" icon.





Та	Tabular Data						
		[A] Solution (Frequency) [Hz]	[B] Solution (Frequency) [Hz]				
1	1.	31.303	31.582				
2	2.	38.797	39.69				
3	3.	62.641	65.477				
4	4.	68.461	69.164				
5	5.	77.279	80.171				
6	6.	77.806	111.06				



Pre-stressed vibration analysis

- Create a pre-stressed modal analysis of 8 holes model
 - Standard Earth Gravity
 - Point Mass 250kg (y = -100mm)

