

Introduction to ANSYS Mechanical

Realize Your Product Promise®



The goal of this workshop is to perform a 4 step analysis on the pipe clamp shown here:

- The bolt will receive a pretension bolt load in LS 1 (locked at LS 2 and for LS3 and 4).
- The pipe will receive an internal pressure during LS 3.
- The pipe will receive an axial force load during LS 4.



ANSYS Project Schematic

Begin a new Workbench session and, from the Project page, choose "Restore Archive . . . " and browse to the file "Pipe_Clamp.wbpz" and Open (location provided by instructor).

When prompted, "Save" using the default name and the same location.

From the "Units" menu verify:

- Project units are set to "Metric (kg, mm, s, °C, mA, N, mV).
- "Display Values in Project Units" is checked (on).



Unit	s Help
	SI (kg,m,s,K,A,N,V)
	Metric (kg,m,s,°C,A,N,V)
-	U.S.Customary (lbm,in,s,°F,A,lbf,V)
~	Metric (kg,mm,s,°C,mA,N,mV)
	U.S.Customary (lbm,ft,s,F,A,lbf,V)
	Display Values as Defined
~	Display Values in Project Units
	Unit Systems



1. From the Static Structural system double click (or RMB > Edit) the "Model" cell

2. When Mechanical opens, verify the units are set to "Metric (mm, kg, s, mV, mA)".



ANSYS Preprocessing

Before we begin preprocessing let's inspect the model as it is currently set up.

- 3. Expand the Connections and Contact branches to view the contacts.
 - Browse the 4 contact pairs. They appear to be scoped correctly but all behavior is currently bonded which we will change (see below).



Contact below the bolt head and between the upper bolt and hole: assumed no separation.

Contact in threaded area: assumed bonded.

5 © 2015 ANSYS, Inc. February 27, 2015





Contact between the clamp and pipe: assumed bonded.



4. Highlight the 1st and 3rd contact regions.

5. In the details change the "Type" to "No Separation".

Our assumption of no separation contact means some sliding can occur in those locations.

Note, the contact between the clamp and the pipe would normally be frictionless in an application like this. However in the interest of time we wish to avoid doing a nonlinear analysis and only demonstrate multi-step solutions. Thus bonded contact is retained.

- 6. Toggle the "Show Mesh" icon to display the mesh.
- Notice a body size control has been added to the bodies. As documented, it is recommended that a more refined mesh be used with bolt pretension.





• Toggle off "Show Mesh" before proceeding.

- 7. Highlight the Analysis Setting branch.
- 8. In the details set Number Of Steps = 4.

Show Mesh					
7. Static Structural (B5)					
Details of "Analysis Settings"					
- Step Controls	•				
Current Step Number 1.	δ.				



9. Bolt Pretension

9. Highlight the cylindrical face on the bolt, RMB > Insert > Bolt Pretension.



- 10. In the tabular data enter a preload of 1 for the first load step (row 1).
- 11. In rows 2, 3 and 4 change the first column to "Lock".
- When complete the table should appear as shown here.
- 12. Highlight the inner surface of the pipe.
- **13.** RMB > Insert > Pressure.





Ta	Ta <mark>bular Data</mark>					
	Steps	Define By	Preload [N]			
1	1.	Load	1.			
2	2.	Lock	N/A			
3	3.	Lock	N/A			
4	4.	Lock	N/A			



ANSYS Solution

- 14. In the tabular data enter a value of 0.1 in row 3 and 0 for row 2.
- When complete, the table should appear as shown below.
 Note, the first 2 rows should default to 0 while the row 4 should maintain the load entered.
- The graph provides a visual confirmation.



14.

Notice in the table there appears to be a duplication of rows. Why? In nonlinear and transient analyses loads are often ramped from a starting to an ending value. Step 1 appears twice in the table to allow ramping from 0 to 1. In static analysis it is meaningless.

9 © 2015 ANSYS, Inc. February 27, 2015



- 15. Highlight the end face of the pipe.
- 16. RMB > Insert > Force.
- 17. In the tabular data enter a value of 10 in row 4 (LS 3).
 - Note, when complete the table should appear as below. It may be necessary to enter a 0 for LS 2 and LS 3.









- **18.** Highlight the cylindrical face of the mounting hole.
- **19.** RMB > Insert > Fixed Support.

20. Solve: this will cause all load steps to be solved sequentially.









 Highlight the Analysis Settings branch and select the graph tab. To display the legend RMB (within the graph window) > Show Legend. This graphically shows the solution processes. Use the "Visibility" section of the details to further configure the display.



8. Request deformation and stress plots to review the solutions. Retrieve different time points to see the complete solution.

ANSYS Go further!

Improve your result accuracy by creating new mesh objects :

- Sizings
- Multizone Method
- Sweep Method
- Mapped Face Meshing

