

Workshop 3B Fluid Pressure Penetration

14.0 Release



Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

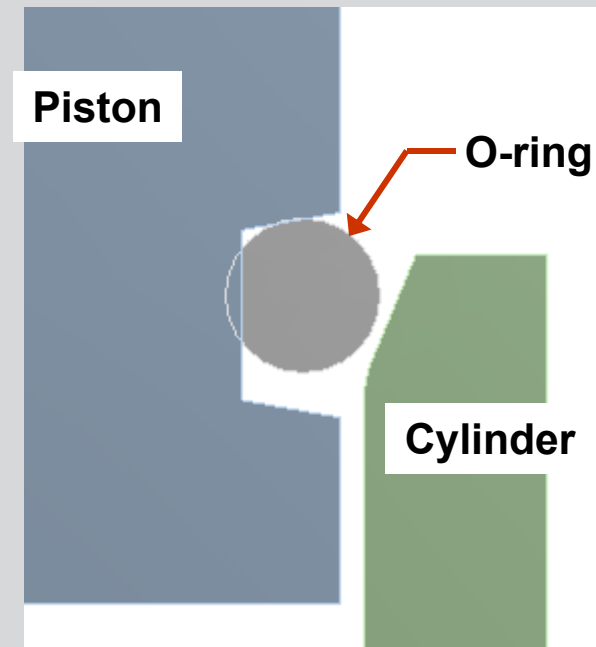
ANSYS Mechanical Advanced Contact & Fasteners

Goal

- Modify Workshop 2C (contact with friction) to include an additional 3rd loadstep that applies fluid pressure to the bottom surface of the O-ring.

Model Description

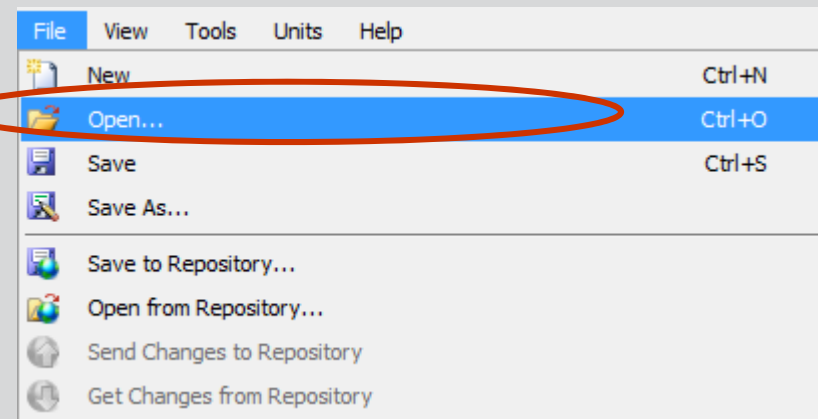
- 2D Axisymmetric
- Piston is constrained from movement.
- Cylinder is displaced to simulate assembly
- Materials:
 - Steel for Piston and Cylinder
 - Elastomer for O-Ring
- 1st load reconciles interference between piston and inside diameter of O-ring
- 2nd load step slides cylinder onto piston-O-ring subassembly.
- **3rd load step apply pressure to the O-ring**



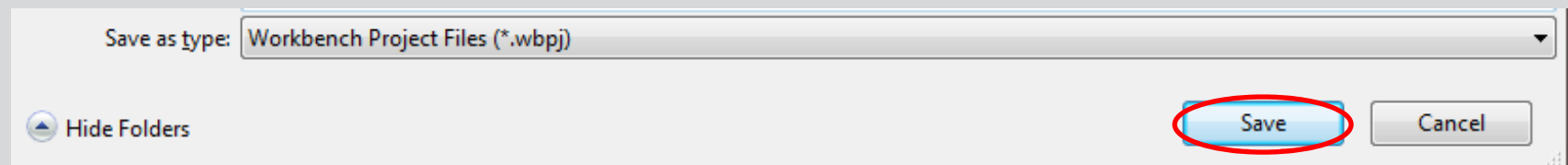
Steps to Follow:

Assuming you have completed and saved workshop 2C , re-open that project

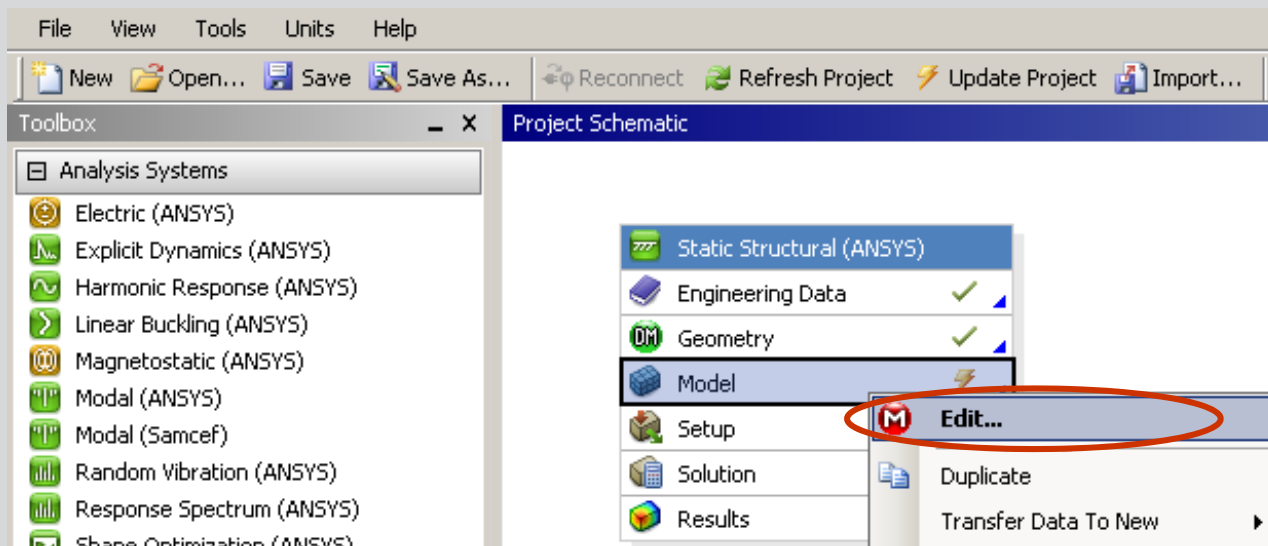
... browse for project file “WS2c_friction.wbpj”

**‘Save-As’ under a new name:**

- File name: “WS3b-pressure”
- Save as type: Workbench Project Files (*.wbpj)



Double click (or RMB=>Edit...) on the Model Cell to open a Mechanical Session



Insert a command object beneath the Frictional O-ring To Piston contact region.

- This will save the contact element type number to a permanent parameter name “cid_1” to be used later

The screenshot displays the ANSYS Workbench interface for a mechanical simulation. The Outline panel on the left shows a tree view of the model structure. Under the 'Frictional - O-ring To Piston' contact, a 'Commands (APDL)' object has been created. A red box highlights this object, and a red arrow points to it. The Commands panel on the right shows the following text:

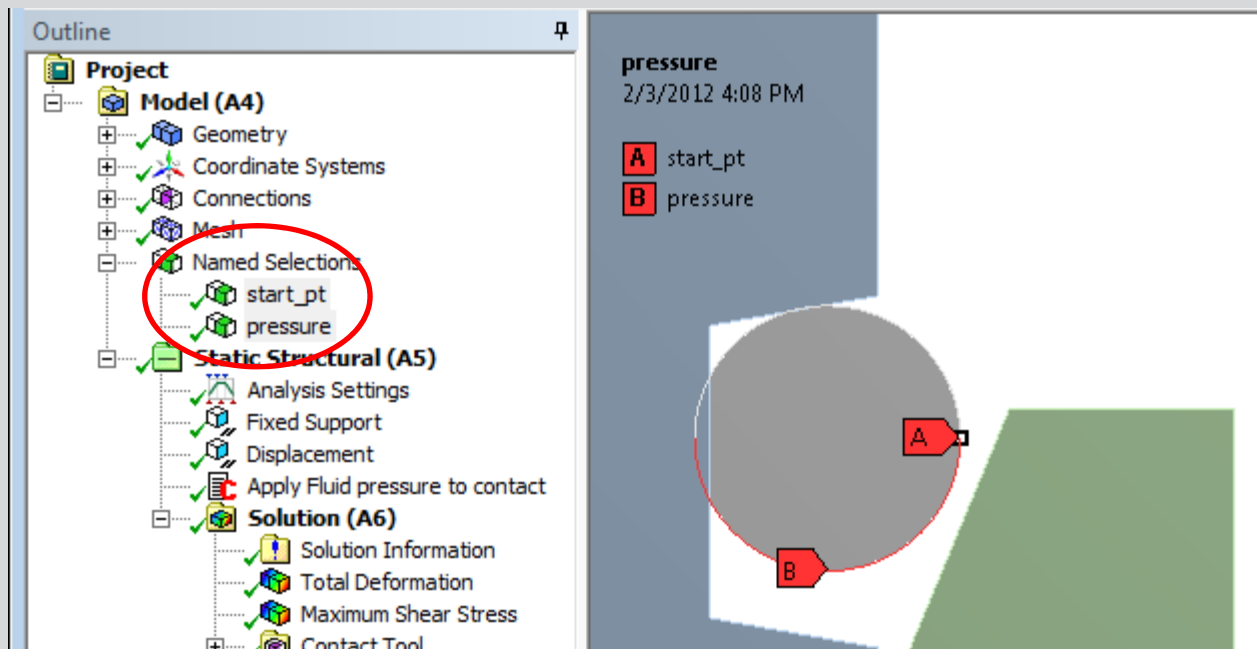
```
! Commands inserted into this file will be executed just after the cont
! The type number for the contact type is equal to the parameter "cid".
! The type number for the target type is equal to the parameter "tid".
! The real and mat number for the asymmetric contact pair is equal to t
! The real and mat number for the symmetric contact pair(if it exists)

! Active UNIT system in Workbench when this object was created: Metric
```

The command `cid_1=cid` is highlighted with a red box in the Commands panel.

Create two Named Selections

- “start_pt”- a single vertex entity that identifies a reference starting point for the pressure load application.
- “pressure”- a single edge along the lower surface of the O-ring that identifies the total potential surface onto which pressure load will be applied.



Highlight the Analysis Settings

- Change number of Steps to '3'
- With current step number set to '3', redefine the auto time stepping specifications as follows:

Details of "Analysis Settings"	
[-] Step Controls	
Number Of Steps	3.
Current Step Number	3.
Step End Time	3. s
Auto Time Stepping	On
Define By	Substeps
Carry Over Time Step	Off
Initial Substeps	25.
Minimum Substeps	5.
Maximum Substeps	1000.
[-] Solver Controls	
Solver Type	Program Controlled
Weak Springs	Off
Large Deflection	On
Inertia Relief	Off
[+] Restart Controls	
[+] Nonlinear Controls	
[+] Output Controls	
Press F1 for Help	

Insert a command object in the Static Structural Environment:

- To execute the necessary select logic on contact elements to receive the pressure load
- To execute the SFE commands needed to apply a pressure load to the lower surface of the O-ring at load step 3.
 - For convenience, these commands are already saved to a text file “WS3b-pressure.inp” in your working directory and can be cut & pasted into this command object.

Outline

Project

- Model (A4)
 - Geometry
 - Coordinate Systems
 - Connections
 - Mesh
 - Named Selections
 - start_pt
 - pressure
 - Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Displacement
 - Apply Fluid pressure to contact
 - Solution (A6)
 - Solution Information

Details of "Apply Fluid pressure to contact"

File	
File Name	
File Status	File not found
Definition	
Suppressed	No
Step Selection Mode	By Number
Step Number	3
Target	Mechanical APDL

Commands

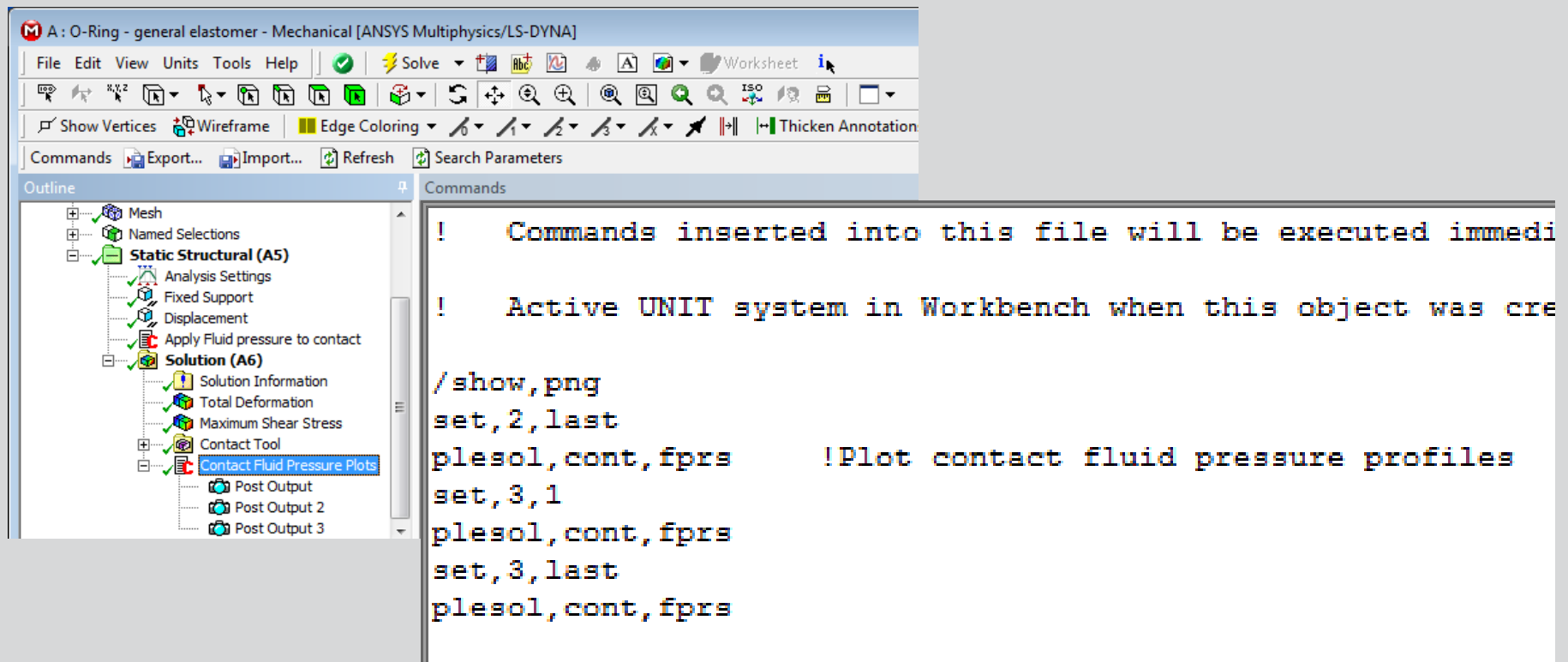
```
! Commands inserted into this file will be executed just prior to the Ansys SOLVE command.
! These commands may supersede command settings set by Workbench.

! Active UNIT system in Workbench when this object was created: Metric (mm, kg, N, s, mV,

cmsel,s,pressure      !Named selection represents all nodes on lower half of seal face
esln,s,1              !select elements attached to nodes
esel,r,type,,cid_1   !reselect only the contact elements with type 'cid_1'
!
sfe,all,1,pres,,50    !apply fluid pressure of 50Mpa to selected contact elements
!
cmsel,s,start_pt      !select starting point
esln
esel,r,type,,cid_1   !reselect only contact element attached to starting point
sfe,all,2,pres,,1    !specify the above location to be initially exposed to fluid pressure
!
allsel                !reselect all elements
```


Optional: A command object can also be added to the Solution branch:

- To visually confirm the correct application of the fluid pressure load in load step 3.
 - For convenience, these commands have been saved to text file 'WS3b-post.inp' in your working directory and can be cut and pasted into this command object.



The screenshot displays the ANSYS Workbench interface for a simulation titled 'A : O-Ring - general elastomer - Mechanical [ANSYS Multiphysics/LS-DYNA]'. The Outline tree on the left shows the following structure:

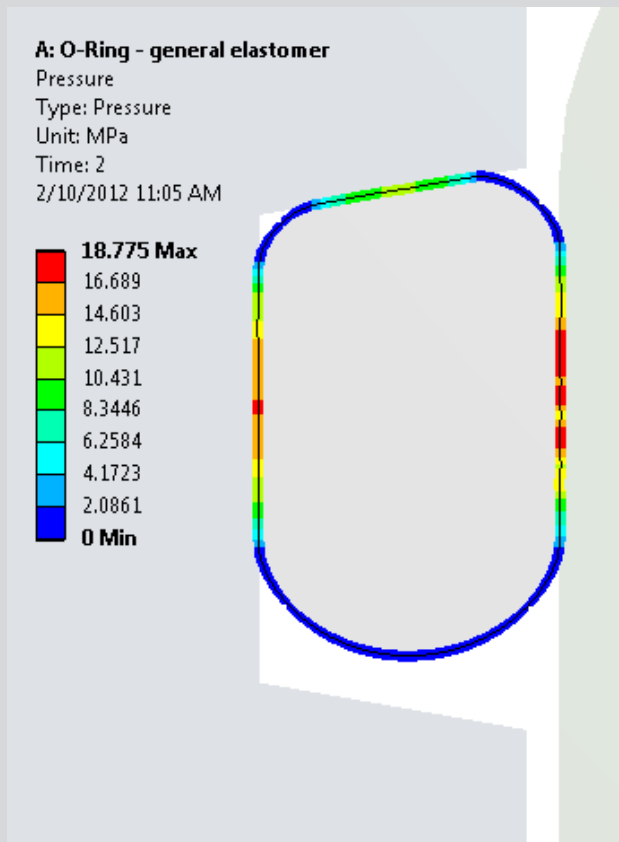
- Mesh
- Named Selections
- Static Structural (A5)
 - Analysis Settings
 - Fixed Support
 - Displacement
 - Apply Fluid pressure to contact
- Solution (A6)
 - Solution Information
 - Total Deformation
 - Maximum Shear Stress
 - Contact Tool
 - Contact Fluid Pressure Plots (highlighted)
 - Post Output
 - Post Output 2
 - Post Output 3

The Commands window on the right contains the following text:

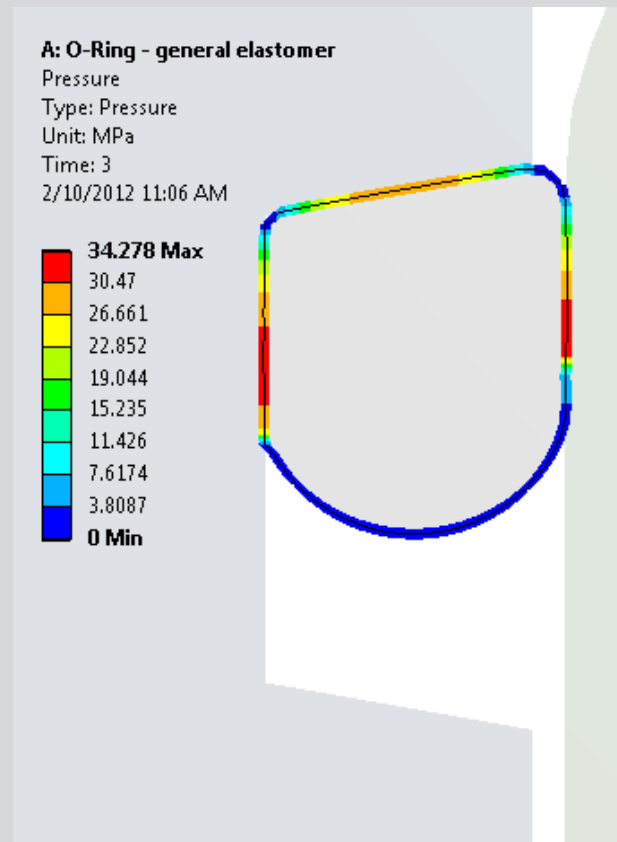
```
! Commands inserted into this file will be executed immedi  
  
! Active UNIT system in Workbench when this object was cre  
  
/show,png  
set,2,last  
plesol,cont,fprs !Plot contact fluid pressure profiles  
set,3,1  
plesol,cont,fprs  
set,3,last  
plesol,cont,fprs
```

Post process results: Compare and contrast LS2 with LS3.

LS 2: Before Fluid Pressure



LS 3: After Fluid Pressure



Expand the command object in the Solution branch to verification of contact fluid pressure loads are applied correctly with MAPDL plots

