

Introduction to ANSYS Mechanical

Realize Your Product Promise®

ANSYS Assumptions

Workshop 3.1 consists of a 2 part assembly representing spur and rack gear components from a 2500 N hand press.

We will solve it as a 2D plane stress model (thickness = 12 mm).







Analysis Goals:

- We are designing a press that should be capable of delivering 2500 N of force in the rack.
- In order to design the mechanism for applying the load we need to know the required torque in the gear to produce the necessary force.
- We'll apply the desired force in the rack and extract the moment reaction at the gear.
- We will use a "Remote Displacement" to constrain the gear (instead of a fixed support) because this type of constraint provides rotational, as well as translational, constraints.



ANSYS Project Schematic

1. Double click "Static Structural" analysis type to add a new system.

2. RMB the Geometry cell and request "Properties".



ANSYS ... Project Schematic

- 3. In the "Analysis Type" field specify "2D".
 - Once this setting is made the properties window may be closed if desired.

Note this setting indicates the model to be analyzed is not a full 3D model but represents a symmetry section. It is important that this is set <u>prior to importing</u> geometry as this setting cannot be changed after the import.

A Property General Component ID Directory Name Notes Notes Jsed Licenses Last Update Used Licenses Geometry Source Geometry File Name	Geometry SYS C:\Users\ir Wechanical	B Value				
Property General Component ID Directory Name Notes Notes Jsed Licenses Last Update Used Licenses Geometry Source Geometry File Name	Geometry SYS C:\Users\ir Wechanical	Value				
Seneral Component ID Directory Name Notes Notes Used Licenses Last Update Used Licenses Seometry Source Geometry File Name	Geometry SYS C:\Users\ir Wechanical	ead \Desktop \Traini				
Component ID Directory Name Notes Notes Jsed Licenses Last Update Used Licenses Geometry Source Geometry File Name	Geometry SYS C:\Users\ir Wechanical	ead \Desktop \Traini				
Directory Name Notes Notes Jsed Licenses Last Update Used Licenses Geometry Source Geometry File Name	SYS C:\Users\irr Wechanical	ead\Desktop\Traini				
Notes Notes Jsed Licenses Last Update Used Licenses Geometry Source Geometry File Name	C:\Users\jre \Wechanical	ead\Desktop\Traini				
Notes Jsed Licenses Last Update Used Licenses Seometry Source Geometry File Name	C:\Users\jra \Wechanical	ead\Desktop\Traini				
Jsed Licenses Last Update Used Licenses Geometry Source Geometry File Name	C:\Users\jre \Mechanical	ad\Desktop\Traini				
Last Update Used Licenses Geometry Source Geometry File Name	C:\Users\jre \Mechanical	ead\Desktop\Traini				
Geometry Source Geometry File Name	C:\Users\jre \Mechanical	ead\Desktop\Traini				
Geometry File Name	C:\Users\jre \Mechanical	ead\Desktop\Traini				
	local_occ_	Geometry File Name C:\Users\jread\Desktop\Training \Wechanical Intro\workshop_input_files \Gear_Set_2D.stp				
Basic Geometry Options						
Solid Bodies						
Surface Bodies						
Line Bodies						
Parameters						
Parameter Key	DS	DS				
Attributes						
Named Selections						
Material Properties						
Advanced Geometry Options						
Analysis Type	2D					
				-		
	Attributes Named Selections Material Properties Advanced Geometry Onlines Analysis Type Ose Associationty	Attributes	Attributes Named Selections Material Properties Managesia Consolver Collings Analysis Type 2D 0SE ASSOCIATIVITY	Attributes Named Selections Material Properties Advanced Geometry Ontions Analysis Type 2D		

ANSYS Geometry Setup

4. From the "Geometry" cell, RMB > "Import Geometry" and browse to: "Gear_Set_2D.stp".



5. Double click the "Model" cell to start Mechanical.



ANSYS Preprocessing

- 6. Set the working unit system:
 - a. "Units > Metric (mm, kg, N, s, mV, mA)".

- 7. Set the Plane Stress options:
 - a. Highlight the "Geometry" branch.
 - b. Verify the "2D Behavior" to be "Plane Stress" (default).







ANSYS ... Preprocessing

- Set the geometry thickness: 8.
 - Highlight the Gear and Rack parts (use shift or **a**. control for multi-select).
 - b. Set the thickness field to 12 mm.

- 9. Set the contact options:
 - **a.** Highlight the contact branch under Connections
 - **b.** Change the contact type to "No Separation".

		Gear		8a.	-		
	De	etails of "Multiple Selecti	on"		д		
	Graphics Properties						
	Definition						
	Suppressed		No				
		Stiffness Behavior		Flexible			
	Coordinate System		Default Coordinate Syst				
		Reference Temperature	By F	invironmen	+		
Rh		Thickness	12.	mm			
·N.		Thickness Mode	war	nuar			



ANSYS ... Project Schematic

- 10. Create a remote point:
 - a. Highlight the Model branch.
 - b. Set the selection filter to "edge select".
 - c. Select the circular inner edge of the gear.
 - d. RMB > Insert > Remote Point.









ANSYS Environment

11. Apply remote displacement on the model:

a. Highlight the Static Structural branch.

b."RMB > Insert > Remote Displacement".

C. Change the scoping method to "Remote Point".

d.Select "Remote Point" from the RP list

e.Set X, Y and Rotation Z = 0.





ANSYS ... Environment

12. Apply frictionless support to the model:
a. Highlight the right edge of the rack.
b. "RMB > Insert > Frictionless Support".





We use a frictionless support along the edge of the rack to simulate the guide the part rides in.

ANSYS ... Environment

13. Apply a force to the model:

a. Select the bottom edge of the Rack.

- **b.**"RMB > Insert > Force".
- **C.** Change to the component method.
- d.Input a Y component = 2500 N.







ANSYS Solution

14. Solve the model.



- **15. Insert a Total Deformation result:**
 - a. Highlight the Solution branch.
 - b. RMB > Insert > Total Deformation
 - c. RMB > Evaluate All Results.



1.	15a.						
<u>.</u>	Insert		Stress Tool	•		15b.	
'Solu /e M	a∏o Rename (F2)	ed Data	Deformation Strain Stress) 	ବ ୍ଷ ଅକ୍ଷ (otal Directional	
finen hent htion	🔄 Open Solver Fi 💱 Worksheet: Re	iles Directory sult Summary	Energy Linearized Stress	•			



ANSYS Postprocessing

- **16. Extract the moment reaction in the gear:**
 - a. Highlight the Solution branch.
 - b. From the context menu choose "Probe > Moment Reaction".
 - c. In the probe details choose "Remote Displacement" from the drop down list.
 - d. RMB > Evaluate All Results.

De	tails of "Moment Rea	ction" P	1	
-	Definition		1	
	Туре	Moment Reaction		
_	Location Method	Boundary Condition	Ц	
	Boundary Condition	Remote Displacement		16c.
	Orientation	Global Coordinate System	Π	
	Suppressed	No	11	
	Options			
	Result Selection	AII		
	Display Time	End Time	Н	
+	Results			
=	Maximum Value Ove	r Time		
	Z Axis		Н	
	Total		Н	
	Minimum Value Over	Time		
	Z Axis			
	Total			
+	Information			



16a.

Solution (A6)

Solution Inform

ANSYS Conclusion

Our stated goal was to determine the required moment that must be applied to the gear in order to produce a 2500 N force in the rack. We conclude a torque of approximately 92,000 N*mm will be required.

Results	
🗆 Z Axis	-92229 N·mm
🗆 Total	92229 N·mm



ANSYS Go further!

If you finish this workshop and find yourself with extra time, you could try the following steps

- **1.** Go into the details box of the created "Remote Displacement";
- 2. Change "Rotation Z" to free;
- 3. Insert "Moment" and scope it to the "Remote Point";
- 4. Give it a value of -92,229 N*mm;
- 5. Delete "Force";
- 6. In the bottom line of the "Rack" put in a "Frictionless Support";
- 7. In the Results section put in a "Force Reaction" probe;
- 8. Observe the value for the "Force Reaction".
 - 16 © 2015 ANSYS, Inc. February 27, 2015