Realize Your Product Promise™

16.0 Release



Workshop 4A: Metal Plasticity

Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

ANSYS Mechanical Introduction to Structural Nonlinearities

Goal:

- Define a nonlinear metal plasticity material for a belleville spring geometry and simulate "spring back" upon application of and subsequent removal of a displacement load.
- Post process stress and strain results
- Generate a force vs. deflection curve on the spring.



ANSYS[®]

... Workshop 4A – Metal Plasticity

2D axisymmetric geometry

- The spring material is a ductile steel sandwiched between two rigid surfaces.
- Frictionless contact is assumed between the spring and the rigid geometries



Steps to Follow:

ANSYS[®]

Restore Archive... browse for file "SNL WS4a-belleville.wbpz"



Save as

- File name: "WS4a-belleville"
- Save as type: Workbench Project Files (*.wbpj)

Save as <u>t</u> ype:	Workbench Project Files (*.wbpj)	•
Aide Folders	Save Cancel	đ

The project Schematic should look like the picture to the right.

ANSYS[®]

- From this Schematic, you can see that Engineering (material) Data and Geometry have already been defined (green check marks).
- It remains to set up and run the FE model in Mechanical
- Open the Engineering Data Cell (highlight and double click OR Right Mouse Button (RMB)>Edit) to verify the linear material properties.
- Verify that the units are in Metric(Tonne,mm,...) system. If not, fix this by clicking on...
 - Utility Menu > Units > Metric(Tonne, mm,...)

Project Schematic



Properti	Properties of Outline Row 3: Structural Steel							
•	А	В	С					
1	Property	Value	Unil					
2	🔁 Density	7.85E-09	tonne mi					
3	🖃 🛛 🔯 Coefficient of Thermal Expansion							
4	Coefficient of Thermal Expansion	1.2E-05	C^-1					
5	🔀 Reference Temperature	22	С					
6	Instropic clasticity							
7	Young's Modulus	2E+05	MPa					
8	Poisson's Ratio	0.3						
9	Alternating Stress Mean Stress	💷 Tabular						
10	Scale	1						

Return to the Project Page.

ANSYS[®]

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



Open the folders beneath the model branch to become familiar with the model set-up.

Highlight "Geometry" and refer to the details window to verify that this is a 2D axisymmetric model.

ANSYS[®]

Inspect the two asymmetric frictionless contact regions on top and bottom of spring which interface with top and bottom rigid boundaries.

Inspect the no-separation contact region which ties down the spring at the bottom corner to prevent rigid body motion during unloading.





- **Review the mesh:** •
 - **RMB>Generate Mesh**

The upper and lower geometries are meshed with one element each, while the belleville spring geometry is a free mesh.





NNSYS[®]

... Workshop 4A – Metal Plasticity

This is going to be a 3 load step analysis:

With the bottom plate fixed:

- LS1: Null Solution (to generate results at origin for force-deflection plot)
- LS2: Apply displacement load (-5mm) to upper plate
- LS3: Remove displacement load
 - Confirm the following Analysis Settings:

Number of Steps:3Weak Springs:OffLarge Deflection:On

For Current Step Number =1, Auto Time Stepping On and with Initial, Minimum and Maximum Substeps = '1'. (Null Solution)

For Current Step Numbers 2 and 3, Program Controlled for Auto Time Stepping.



De	Details of "Analysis Settings"							
Ξ	Step Controls							
	Number Of Steps	3.						
	Current Step Number	1.						
	Step End Time	1. s						
	Auto Time Stepping	On						
	Define By	Substeps						
	Initial Substeps	1.						
	Minimum Substeps	1.						
	Maximum Substeps	1.						
Ξ	Solver Controls							
	Solver Type	Program Controlled						
	Weak Springs	Off						
	Large Deflection	On						
	Inertia Relief	Off						

Review the predefined displacement load on the upper plate for the • three load steps.





Execute Solve:

🔣 Solve

After solution is complete, review convergence history:



- Post Process results at Load step 2:
 - Note how high the stress in the spring is at the end of LS2.
 Recall, this is still linear elastic material.
 - At LS3 (not shown), plastic strain is zero and there is no permanent deformation of the spring upon unloading as expected.



- Generate Force vs. Deflection Curve of Spring
 - With Solution Branch Highlighted:
 RMB>Insert>User Defined Result
 - Scope result to the upper rigid plate
 - Define the expression as 'abs(FY)' for absolute value of force in Y-direction



User Defined Result							
etails of "User Defined Result"							
Scope							
Scoping Method	Geometry Selection						
Geometry	1 Face						
Definition							
Туре	User Defined Result						
Expression	= abs(FY)						
Input Unit System	Metric (mm, kg, N, s, mV, mA)						
Output Unit	Force						
Ву	Time						
Display Time	Last						
Coordinate System	Global Coordinate System						
Calculate Time History	Yes						
Use Average	Yes						
Identifier							
= Results							
Minimum							
Maximum							
Information	· · · · · · · · · · · · · · · · · · ·						

Repeat Procedure for Displacement in 'UY'

ANSYS[®]

De	Details of "abs(UY)"							
	Scope							
	Scoping Method	Geometry Selection						
	Geometry	1 Face						
	Definition							
	Туре	User Defined Result						
K	Expression	= abs(UY)						
	Input Unit System	Metric (mm, kg, N, s, mV, mA)						
K	Output Unit	Length						
	Ву	Time						
	Display Time	Last						
	Coordinate System	Global Coordinate System						
	Calculate Time History	Yes						
	Identifier							
	Suppressed	No						

Fixe	d Support	√ G
	Solut Insert 🕨	Stress Tool
	Direc Equiv allo Rename Equivalent Elastic Strain	Deformation Strain Stress Linearized Stress
Refir	ement	Fatigue
Loops	1.	Contact Tool
	2.	Drebe
	(See User Defined Result
		Commands

- Highlight both User Defined Results:

RMB>Rename based on Definition

- Highlight Solution Branch:

RMB>Evaluate results



Insert a Chart Tool for plotting FY vs UY



- Fill in Chart tool Details Window as Follows:
 - Definition: Select 'abs(FY)' and 'abs(UY)' from Solution Branch
 - Chart Tool: X Axis: abs(UY) (Max)
 - Axis Labels:
 - X-Axis Label: Deflection
 - Y-Axis Label: Force

- Input & Output Quantities:
 - Omit: Time, abs(FY)(Min), abs(UY)(Min)



Resulting Chart of Force vs Displacement for linear material is a straight line with no permanent deformation as expected



Duplicate the Static Analysis

- Return to the Project Schematic
- Highlight the Model Cell and RMB> Duplicate

Toolbox _	× Project S	Schem	atic		
🖃 Analysis Systems					
🙆 Electric (ANSYS)	- I.				
💹 Explicit Dynamics (ANSYS)		•	A		
🔯 Harmonic Response (ANSYS)		1 🚾	Static Structural (ANS)	/S)	
😥 Linear Buckling (ANSYS)		2 🦪	🔰 Engineering Data		
🔟 Magnetostatic (ANSYS)		3 🕅	Geometry		
🎹 Modal (ANSYS)		4	Madal (1
🎬 Modal (Samcef)		- •		6	Edit
📶 Random Vibration (ANSYS)		5 🦉	👢 Setup		
📶 Response Spectrum (ANSYS)	1	6 🧃	Solution	48	Duplicate
Shape Optimization (ANSYS)		7 🥳	Results		Transfer Data To New
🤓 Static Structural (ABAQUS)			Alleville Caving Linear Ma	4	Lodate
🧒 Static Structural (ANSYS)			elleville opril ig-cil lear ma	-	opuace
🤓 Static Structural (Samcef)				12	Refresh
🚺 Steady-State Thermal (ANSYS)					Clear Generated Data
🔁 Thermal-Electric (ANSYS)					Reset
🚾 Transient Structural (ANSYS)				āb	Rename
🚾 Transient Structural (MBD)					Ronano B. U
🔃 Transient Thermal (ANSYS)					Properties

Disassociate material properties link

- The second analysis is going to be with metal plasticity defined
- Highlight the Engineering data link and RMB>Delete

Project	Sch	iemat	tic						
	•		А			•		В	
	1	_	Static Structural (ANSYS)			1	_	Static Structural (ANSYS)	
	2	٢	Engineering Data	~ <				Engineering Data	∼ .
	3	00	Geometry	× .	×	De	elete		 _
	4	۲	Model	 Image: A second s		Pr	oper	ties	 Image: A second s
	5	٢	Setup	 Image: A second s		5		Setup	7
	6	1	Solution	 Image: A second s		6	(iii)	Solution	4
	7	@	Results	 Image: A second s		7	۲	Results	1
		Be	lleville Spring-Linear Materia	ls		Co	py ol	f Belleville Spring-Linear Mal	terials

Project Schematic should now look like the diagram below

- We can now modify the Engineering data in Table B without effecting the model and/or results of Table A.
- Change the title of the new Analysis to: "Belleville Spring-Nonlinear Materials"
- Open the Engineering Data Cell in Table B



Insert a Metal Plasticity Model

- From the Tool Box, open the Plasticity Folder
- Highlight Multilinear Isotropic Hardening and RMB>"Include Property"
- The new material should now appear in the Properties dialogue box

			Properti	es of Outlir	ne Row 3: Structural Steel		▼ +⊐	x
			•		A	В	С	D
🔨 WS5a-belleville - Workbench			1	Prope	Property		Unit	6
File View Tools Units Help			<u> </u>	0-				
📔 New 对 Open 🛃 Save 🔣 Save As	l∉φ Reco	nnei	2	12	Density	7.85E-09	tonne mm^-3	L
Toolbox _ ×	Outline	Filte	3	🗉 ն	Coefficient of Thermal Expansion			
	-		6	😐 🎦	Isotropic Elasticity			F
Elinear Elastic	1							님
Experimental Stress Strain Data ■	2	٢	9	2 📶	Multilinear Isotropic Hardening	💷 Tabular		
Hyperelastic	3		12	H 17	Alternating Stress Mean Stress	Tabular		F
Plasticity	4							╠╝
🔁 Bilinear Isotropic Hardening			14	🛨 🔁	Strain-Life Parameters			
Multilinear Isotropic Hardening	e Property		22	62	Tencile Vield Strength	250	MDa	
Pilinear Kinematic Hardening			22		rensile held barengan	230	inca	
Multilinear Kinematic Hardening Expan	id All		23	1	Compressive Yield Strength	250	MPa	
🖽 Life Collap	se All							듣
in character in the second sec		24	12	Tensile Ultimate Strength	460	MPa		
			25	1	Compressive Ultimate Strength	0	MPa	



Define Plasticity data

 Fill in plastic strain and stress data as shown to the right

	able of Properties Row 12: Multilinear Isotropic Hardening							
		А			В	с		
Ī	1	Temperature (C) 📮		1	Plastic Strain (m m^-1) 🗦	Stress (MPa) 💌		
	2	22		2	0	250		
	*			3	0.001	265		
			·	4	0.002	275		
				5	0.003	278		
				6	0.004	279		
				*				
							1	

2

🚽 Д

- From the Utility Menu, read in the modified material properties with
- hart of Properties Row 12: Multilinear Isotropic Hardening

- Return to Project Workspace
 - Refresh Project





From the project schematic, highlight and open the model cell in Table B.



All the geometry entities, meshing specs, boundary conditions, loads and analysis settings are preserved from the previous analysis.

- Execute the Solve with the newly defined plasticity properties



ANSYS[®]

... Workshop 4A – Metal Plasticity

Return to Analysis Settings. For LS 2 and 3 specify the following for autotime stepping:

	Static Structural (B5) Static Structural (B5) Analysis Settings Displacement Fixed Support Solution (B6)					
I	Details of "Analysis Settings"					
I		Step Controls				
I		Number Of Steps	3.			
I		Current Step Number	2.			
I		Step End Time	2. s			
I		Auto Time Stepping	On			
		Define By	Substeps			
I		Carry Over Time Step	Off			
I		Initial Substeps	5.			
		Minimum Substeps	3.			
		Maximum Substeps	10.			
ľ						

Execute Solve...

🖻 B : Copy of Belleville Spring-Linear Materials ·	Mechanical [A	NSYS Multiphysics]
] File Edit View Units Tools Help 🗍 🥑 🕶	💖 Solve 👻	🗸 Show Errors 🏥 📷 🖄 🛷 🔺 📝 🗣 🎲 Worksheet 🗼 🛒 🎀 🦎 🦒 🗸
ل ⊐ Show Vertices 🖧 Wireframe 🗳 Show Mesh	🙏 🚦 Rand	om Colors 🔣 Annotation Preferences
Environment 🔍 Inertial 🗸 🖓 Loads 🗸 🖓 Support	s 👻 🔍 Condition	ns 👻 🖤 Direct FE 👻 📄
Joint Configure Configure Assemble Δ=	0	Set Revert

- After solution is complete, review Solution output:
 - Confirm that the metal plasticity, as defined, was included in this new run

PLASTIC	(PLAS) Table	For	Material 3
ISOTROPIC HARDENING PLASTICITY			
Temperature = 22.000000			
Point	PlStra	in	Stress
1	0.000000e	+000	2.500000e+002
2	1.000000e	-003	2.650000e+002
3	2.000000e	-003	2.750000e+002
4	3.000000e	-003	2.780000e+002
5	4.000000e	-003	2.790000e+002

*** MAX PLASTIC STRAIN STEP = 0.3968E-02 CRITERION = 0.1500 *** AUTO TIME STEP: NEXT TIME INC = 0.30000 INCREASED (FACTOR = 1.5000)

• Review the Convergence History. Compare this with the Linear material run.



- Post Process results at Load step 2 as before:
 - Compare the max stress in this material with the linear material



 Note also that the spring now takes a permanent set after load is removed as expected.



- Highlight the Chart tool and Plot Force vs Deflection as before.
 - Note the nonlinear path of the curve reflecting the influence of the material yielding and taking a permanent set.
 - Note also the difference in the magnitude of the load required to produce the same deflection with this material verses the linear material, underscoring the importance of considering nonlinear material behavior in some designs.



• To improve the quality of the force deflection curve, try rerunning the analysis using a minimum of 15 substeps for LS 2 and 3.



ANSYS°