Mechanics of Thin-walled Structures CONICAL



PROBLEM DESCRIPTION



Static, linear analysis of thin-walled aluminum skin, reinforced by a stiffening ring, will be performed.

In the course of this exercise, students will gain basic knowledge about: **creating geometry and mesh; applying loads and boundary conditions; setting up static, linear analysis** and **results post-processing.**

Two cases will be considered:

- 1. When stiffening ring is (relatively) rigid
- 2. When stiffening ring is made of **steel** (and its deformations influence the deformation of the aluminum skin)

Units: mm, N, MPa

÷ Fil	le Group Viewport	Viewing	Display Preferen	ces Tools Help Utilit	ies			
	New	Ctrl+N	me Geometry	Properties Loads/	BCs Meshing Ar	nalysis Results		
d	Open	Ctrl+O						
	Open Recent	Ctrl+R	£€₽₽₽			1/2 🔊 🎽 👪		
	Close	Ctrl+W	🞯 🖪 Q Q	🔁 🕾 🔺 🥔		L 🔧 °; 🖷		
	Save	Ctrl+S	Viewport	Display	Orientation	Misc.	Web Model Tree	
	Save a Copy							
	Utilities	+	Template Da	tabase Name				
	Import		C:\MSC.So	ftware\PATRAN\2010/temp	late.db		Mod	del Preference for:
	Export		-					ex_1.db
	SimManager	•		Chang	je Template		Tol	erance
			L. Madife Dec	<i></i>				Based on Model
	Session	•	I♥ Modify Pre	rerences				
	Print		Set Workin	t Working Directory to Database Location				bal Model Tolerance:
	Images		-					0.005
	Report		Look <u>i</u> n:	🍌 mts				
	Quit	Ctrl+Q	Name	^	Date m	odified Ty	Ana	alysis Code:
				No items n	natch your search.		MS	C.Nastran
							e Ana	alysis Type:
							Str	ructural 🔻
			4			•		OK Reset
			Classical State			011		
			File <u>n</u> ame:	jex_1.db		OK		
	Create a new data	abase:	Files of type	Database Files {*.db}	-	Cancel		
	a. File / New							
	b. Enter ex_1.	db as						
	the File nam	ne						
	c. Click OK							

3

d. Select Default

e. Click **OK**

File Group Viewpo	rt Viewing	Display d eren	ces To	ools Help Utili	ties					
🗋 🖻 🗠 🖱 💰 🗖	🕀 🕈 Hom	ne Geometry	Prop	perties Loads,	/BCs	Meshing A	nalysis	Results		
□ ≥ ≥ ∎ ⊕	88 89 6	₹ + + +	B	7 🗭 🌐	Ž.x	ׯ Ż Ż X Ż	1	7		
<u>∎ ⊳ 0 ≼ ≶</u>	6+6	୭ 册 ୧ ୧	e i e	14 KR 🔺 🤣	ZX	z, ^V x ^V , ^x ² , ^y x ^x , ^y z	L °>	°∖∎∖		
Defaults	Transforms	Viewport	с	Display	а	Orientation	b 📙	ic.	Web	Model Tree

4

Create geometry points:

- a. Click on the Right side view icon
- b. Click on the **Point size** icon
- c. Click on the Cycle Background icon
- d. Click on the Geometry icon
- e. Geometry: Create/Point/XYZ
- f. Uncheck Auto Execute
- g. Enter **[0 225 0]** as the Point Coordinates List
- h. Click Apply

7

i. Create two more points using coordinates: [0 100 770], [0 144 495]



Auto-Execute Option

The Auto-Execute option will automatically apply the operation when the last field is filled in.











Delete Original Elements

g

Base Entity List

h

-Apply-

f







-Apply-







File Group Viewport Viewing Display Preferences	b Help Utilities roperties Loads/BCs Meshing Analysis Results	
D ≧ ≧ B B S S Q ⊕ ⊕ ⊕ D C B S S S C Q ⊕ ⊕ ⊕ (S) Defat Defat a	Image: Second system Image: Second system <td< th=""><th>Action: Create</th></td<>	Action: Create
	Constitutive Model: Linear Elastic	Object: Isotropic
	Property Name Value	
 a. Click on the Reset graphics icon Define materials: b. Click on the Properties tab c. Click on Isotropic icon, then: Create/Isotropic/Manual Input d. Enter aluminum as the Material Name e. Click Input Properties f. Enter 73000 as Elastic Modulus and 0.33 as Poisson Ratio 	Elastic Modulus = 73000 Poisson Ratio = .33 Shear Modulus = Density = Thermal Expan. Coeff = Structural Damping Coeff = Reference Temperature =	Existing Materials
 h. Click Apply i. Repeat steps f+k to define the second, relatively stiff material e.g.: ring_mat, E = 2e7, v = 0.29 	Current Constitutive Models:	Material Name aluminum d Description
	OK g Clear Cancel	Input Properties e Change Material Status

h	Apply

File Group Viewport Vie	wing Display Prefere	nces a Help Utili	ties		
🕒 🖻 🗠 🖱 🏑 🗭 🕫	Home Geometry	Properties Loads,	/BCs Meshing Ai	nalysis Results	
□ ≌ 달 물 물 중	9 9 8 + +	87.		>>>>>	
≞⊵₿∡≶	§ 👰 Ħ Q Q	€₽≈≈△∅		L 😵 😘 🖺	
Defaults Transf	orms Viewport	Display	Orientation	Misc.	Web Model Tree

💷 Select Material 📃 🗖	X
Select Existing Material	
aluminum ring mat f	^
	v
Filter *	
External Materials	
Cancel	

Assign the properties (shell):

	Stan Homogeneous Plate(COLLA	D4)			Action: Create	<u>'</u>
Select Material	Property Name	Value	Value Type		Object: 2D 🔻	b
Select Existing Material	Material Name	m:aluminum	Mat Prop Name	<u>^</u>	Type: Shell 🔻]
aluminum ring_mat f	[Material Orientation] Thickness		CID V Real Scalar V		Sets By: Name 🔻	
<u>ب</u> ح	[Nonstructural Mass]	8	Real Scalar			
	[Plate Offset]		Real Scalar	=		
	[Fiber Dist. 1]		Real Scalar		•	
Filter *	[Fiber Dist. 2]		Real Scalar		Filter	ż
External Materials	[Nonlinear Formulation(SOL400)]		String 🔻		Property Set Name	
Cancel	•		4	*		
sign the properties (shell):					Options:	-1
a. Double-click Properties tab	Enter the Thickness or colority f	ald with the isse. Consify Element N	ladal by aslactics it from the		Standard Formulation	
b. Properties: Create/2D/Shell c. Enter shell as the New Set	dropdown.	eid with the icon. Specify Element i	lodal by selecting it from the	<u> </u>		
Name				-	Input Prop	perties
d. Click Input Properties e Click on the Mat Prop Name					Select Applica	ition Region
icon	<u>ок</u> h	Clear	Cancel			
f. Select aluminum		16			Ap	ply
S. LINCE I as the HINCKHESS						

'n

b.

d

h. Click OK



	File Group Viewport	Viewi	ng Display Preferer	nces Tools Help	Utilities					
	🗅 🖻 🗠 🖱 💰 🎑 🛱	₹ ₹	Home Geometry	Properties I	Loads/BCs	Meshing A	nalysis	Results		
	D C C C C C C C C C C C C C C C C C C C	हिं दिन्द्र Starting	Image: Constraint of the second state Image: Constraint of the second state <td>(2) (2) (2) (2) (2) (2) (2) (2) (2) (2)</td> <td></td> <td>$\begin{array}{c} x \overset{g}{\underline{z}} \end{array} \left[\begin{matrix} \overline{y} & x \\ \overline{y} & x \end{matrix} \right] \\ z \overset{g}{\overline{y}} & \overset{g}{\overline{y}} & \overset{g}{\overline{z}} & x \end{matrix} \right] \\ x \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} \\ x \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} \\ \hline \text{Orientation} \end{array}$</td> <td>C 20</td> <td> ↓↓ </td> <td>La Model Tree</td> <td></td>	(2) (2) (2) (2) (2) (2) (2) (2) (2) (2)		$\begin{array}{c} x \overset{g}{\underline{z}} \end{array} \left[\begin{matrix} \overline{y} & x \\ \overline{y} & x \end{matrix} \right] \\ z \overset{g}{\overline{y}} & \overset{g}{\overline{y}} & \overset{g}{\overline{z}} & x \end{matrix} \right] \\ x \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} \\ x \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} & \overset{g}{\overline{z}} \\ \hline \text{Orientation} \end{array}$	C 20	 ↓↓ 	La Model Tree	
		Ge	neral Section Beam (CBAR	.)				Action: C	reate 🔻	
		Pro	perty Name	Value		Value Type		Object: 1	D 🔻	а
		[Se	action Name]	na:		Properties 🔻	Τ	Туре: В	eam 🔻	
		Ма	iterial Name			Mat Prop Name	XX =		1	
		В	ar Orientation			Vector 🔻		Sets By: Na	me 🔻	
Assign the	properties (ring):		Offset @ Node 1]			Vector				
a. Prope	erties:		Offset @ Node 2]			Vector				-
Creat	e/1D/Beam	[Pir	nned DOFs @ Node 1]			String 🔻		•		F.
b. Enter Name	ring as the New Set	[Pir	nned DOFs @ Node 2]			String 🔻		Filter	*	
c. Click	- Input Properties	Ar ∢	ea			Real Scalar	- mi	Property Se	et Name	
d. Click	on the Create	p		Create	Sections			ring		b
Jeth				ТГ		d		Options:		
				Beam	Library	-		General Sec	tion 🔻	
				Assoc. B	leam Section			Standard Fo	rmulation	•
		Ent to c	er the Section Name, select create a new section.	t existing section using th	ie icon, or use t	he create sections ico	n below 🔺			
								Ir	put Properties	С
							Ŧ	Selec	Application R	egion
			ок	Cle	ar	Canc	el			
									Apply	



File Group Viewport	Viewing Display Preferences Tools Help Utilit	ties	
🕒 🖻 🗠 🖱 🏑 🗖	F → Home Geometry Properties Loads/	/BCs Meshing Analysis F	Results
	S S R + + + R R + H R R A R R A R A R A R A R A R A R A R		
	General Section Beam (CBAR)		Action: Create 🔻
	Property Name Value	Value Type	Object: 1D 🔻
aluminum	[Section Name] section	Dimensions	Type: Beam 🔻
ring_mat	Material Name m:steel	Mat Prop Name k 🐹 🗉	
✓	Bar Orientation <0 0 1> m	Vector 🔻	Sets By: Name
	[Offset @ Node 1]	Vector	shell
	[Offset @ Node 2]	Vector	
Filter *	[Pinned DOFs @ Node 1]	String 🔻	▼
External Materials	[Pinned DOFs @ Node 2]	String 🔻	Filter *
Cancel	Area 24.	Real Scalar T	Property Set Name
	Create Sectio	ns	ring
	ТГІ		Options:
k. Click on the Mat Prop	Beam Librar	У	General Section
I. Select ring mat	Assoc, Beam Se	ction	Standard Formulation
m. Enter <0 0 1> for the Bar	Enter the Bar Orientation or select a field with the icon. Specify with the dropdown.	y if this value is a Vector or Node ID 🔺	
Orientation			Input Properties
		Ŧ	Select Application Region
	OK Clear	Cancel	L
			Apply



Erile Group Viewport Viewing	Display Preferences Tools Help Util Ome Geometry Properties Loads	ities a /BCs Meshing Analysis Results	
Defaults	⊕ ⊕ ⊕	Image: Second state Image: Second state<	Action: Analyze Object: Entire Model b Method: Analysis Deck
□ OP2 ▼ XDB g Print □ Punch □ MASTER Only □ MASTER/DBALL	Static Solution Parameters	MSC.Nastran Solution Type	Code: MSC.Nastran Type: Structural
h OK Defaults Cancel	Inertia Relief Alternate Reduction SOL 600 Run SOL 700 Run	LINEAR STATIC NONLINEAR STATIC NORMAL MODES BUCKLING	Available Jobs
Run a linear analysis: a. Click on the Analysis tab	Contact Parameters Shell Normal Tol. Angle = Mass Calculation: Lumped	C COMPLEX EIGENVALUE C FREQUENCY RESPONSE C TRANSIENT RESPONSE C NONLINEAR TRANSIENT	Job Name ex_1 Job Description (TITLE)
 d. Select LINEAR STATIC as the Solution Type 	Data Deck Echo: None Plate Rz Stiffness Factor = 100.0 Maximum Printed Lines =	IMPLICIT NONLINEAR DDAM Solution Select ASET/QSET	SUBTITLE
e. Click Solution Parameters f. Click Results Output Format g. Uncheck Print	Wt-Mass Conversion = 1.0 Node i.d. for Wt. Gener. =	Solution Parameters e	LABEL Translation Parameters
i. Click OK j. Click OK k. Click Apply	Default Load Temperature = Image: Complexity of the second seco	Solution Sequence: 101	Solution Type Direct Text Input Select Superelements Subcases
	OK Defaults Cancel	OK j Cancel	Subcase Select



File Group Viewport Viewing Disp	lay Preferences Tools Help Utili	ties		
🕒 🖻 🗠 🖱 💰 🎑 🖶 = Home	Geometry Properties Loads	/BCs Meshing Analysis	Resul Action:	Access Results 🔻
🗅 🖻 🖉 🖉 🖉 😫	\$++ + + + + + + + + + + + + + + + + + +		🔊 🔊 🖁 Object:	Attach XDB 🔻 a
<u></u>	▋�� ₽₽≈≈△∅		୍ଦ୍ଧ 🍳 🖁 Method:	Result Entities 🔻
Defaults Transforms V	iewport Display	Orientation	Misc.	
			Code:	MSC.Nastran
			Туре:	Structural
Attach the results file, when the analysis]			Ral
job is completed:			ex_1	e Jobs 🔄
a. Analysis: Access Results/Attach				-
XDB/Result Entities			٠	ł
D. CICK Select Results File			Job Nor	_
d. Click Apply			lex 1	
			Job Des	cription (TITLE)
			MSC.N	astran job
				-
			-	
			SUBTITL	E
			LABEL	
			,	
				Select Results File
				Translation Parameters
	23			Apply d

File Group Viewpo	rt Viewing D	isplay Preferen	ces Tools Help Utilit	ies		b	
🗅 🖻 🗠 🖱 💰 🗖	😫 🕫 Home	Geometry	Properties Loads/	BCs Meshing A	nalysis Result	ts	
□ ≝ ≝ 🛢 🖨	88 89 😫	•++	Ø 🗇 🗰 📰		~ @ ? !	3	F •
≞⊵⊜∡≶	6+64 🗯	₽₽₽₽			L 😵 a		
Defaults	Transforms	Viewport	Display	Orientation	Misc.	Web	Model Tre



Post-process the results:

- a. Click on the Plot/Erase Geometry icon
- b. Click on the Results tab
- c. Results: Create/Quick Plot
- d. Select Result Cases: Default, A1:Static Subcase
- e. Select Fringe Result: Displacements, Translational
- f. Quantity: Y Component
- g. Select Deformation Result: Displacements, Translational

24

- h. Click on the Deform Attributes icon
- i. Enter 0.04 as the Scale Factor
- j. Uncheck Show Undeformed
- k. Click Apply

<u>Remark</u>: To capture the plot use File / Images...

(Web) [Model free]	Action: Create 🔻
Action: Create	Object: Quick Plot 🔻
Object: Quick Plot	
	Select Result Cases
	Default, A1:Static Subcase;-MS
Show Viewport Legend	d
Deformed:	
Render Style: Wireframe 🔻	
Line Style:	
Line Width:	<
Scale Interpretation	Select Fringe Result
Model Scale C True Scale	Bar Stresses, Maximum Combin
	Bar Stresses, Minimum Combine
Scale Factor 0.04	Displacements Translational
	Stress Tensor,
Show Undeformed	•
j	Quantity: Y Component
	Select Deformation Result
	Constraint Forces, Translational
Title Editor	Displacements, Translational
Show Title Lock Title	
Show Maximum Label	•
Label Style	Animate
Apply k Reset	Apply

A

е

g

þ.

File Group Viewpo	ort Viewing Display	a ences Tool	s Help Utilities				
🕒 🖻 🗠 🖱 💰 🗖	+ 🔁 ≠ Home	Geometry Proper	ties Loads/BCs	Meshing Ana	alysis Results		
Dĕĕ∎⊜	88 81 81 4	•+++	🔹 🔠 😰	xž VX VX X	/ 🔊 🕫 👪		
🖻 🗠 🖱 🏑 💉	8 8 D 🖬	99	SB 🖉 🗾	z, ^v _x ^v ^x _z ² ^v _x ^x , ^v _z	L 💊 °; 🖺		
Defaults	Transforms Vie	ewport D	isplay	Orientation	Misc.	Web	Model Tree

Create a cylindrical coordinate system:

- a. Click on the **Geometry** tab
- b. Geometry: Create/Coord/3Point
- c. Type: Cylindrical
- d. Uncheck Auto Execute
- e. Click Apply



	Action:	Create 🔻	
	Object:	Coord 🔻	b
	Method:	3Point 🔻	
	Coord ID Lis	st	
	1		
	Туре:	Cylindrical	▼ C
	Refer. Coo	rdinate Frame	
	Coord 0		
d	🗌 Auto Exe	cute	
	Origin		
	[0 0 0]		
	Point on A	kis 3	
	[0 0 1]		
	Point on Pla	ane 1-3	
	[1 0 0]		
	e	-Apply-	

Г



g

26



Post-process the results:

- a. Click on the Results tab
- b. Results: Create/Fringe
- c. Select Result Cases: Default, A1:Static Subcase
- d. Select Fringe Result: Stress Tensor
- e. Quantity: Z Component (axial stress)
- f. Click on the Plot Options icon
- g. Select CID
- h. Select the cylindrical coordinate system
- i. Click Apply









Object: Graph ▼ Method: Y vs r
Select Result Cases
Y: Result
Bar Stresses, Maximum Combined Bar Stresses, Minimum Combined Constraint Forces, Translational Displacements, Translational Stress Tensor.
Position(At Z1)
Quantity: Z Component 🔻
X: Path Length 🔻
Apply Reset

Action: Create 🔻



Compare obtained graph with vector results:

- a. Click on the Reset graphics icon
- b. Select Right side view
- c. Fit window
- d. Click on Results/Create/Marker/Vector
- e. Select **Constraint Forces**, **Translational** for the Vector Result
- f. Change option to **Component** and toggle on only **ZZ** option
- g. Click on Display Attributes
- h. Toggle **Constant** vector color option and choose color for ZZ component
- i. Change Length to Model Scaled
- j. Toggle off Show Vector Label

k. Click Apply

	X X Z
b Orientation	n
Results	
Action: Create	
Object: Marker 💌 d	
Method: Vector 💌	
	S.
Select Result Cas	≣Ţ
Default, A1:Static subcase	
<	
Select Vector Result	
Bar Stresses, Axial Bar Stresses, Bending	
Constraint Forces, Rotational	
Displacements, Rotational	
Show As: Component	f
□ Y+Z □ Z+X □ X+Y □ X+Y+Z □ X+Y+Z	-
Animate	

Misc. Web Model Tree
Results
Action: Create
Object: Marker
Method: Vector 🔻
O Spectrum O Constant
Y+Z: Z+X: X+Y:
Show Viewport Legend
Vector definition
Length: Model Scaled
Scale Factor 0.1
Anchor Style:
Head Style:
Line Style:
Title Editor
Show Title Lock Title
Show Max/Min Label
Show Vector Label
Label Style
Show on Deformed
Show on Deformed



File Group Viewport Viewing	Display Preferen	ices Tools Help Utilit	ties			
🕒 🖻 🗠 🖱 🏑 🗛 🖶 🕫 Ha	ome Geometry	Properties Loads/	BCs Meshing A	nalysis Results		
∟≝≝∎⊴ ∜∛	()	\$\$\$		/ / / / / / / / /	8	Fe
≞⊵₿ፈ≶ ≶⊗		∎∎∎≣™≏⊘		LNI		
Defaults Transforms	Viewport	Display	Orientation	Misc.	Web	Model Tree

ANALYSIS NO. 2

- change of the material properties

Change ring material properties, run second analysis and compare results with recently obtained:

- a. Go to Properties tab and select Modify/Isotropic
- b. Choose ring_mat
- c. Change Young's Modulus to **2e5** (note that previous material was 100 times stiffer!)
- d. Click OK
- e. Click Apply
- f. In the **Analysis** tab, select Analyze/Entire Model/Full Run
- g. Change Job Name to ex_1b
- h. Select Apply
- i. After analysis is done, attach ex_1b.xdb file and plot results in the same manner as previously. Results refering to the second analysis will be tagged as A2: Static subcase

			Arabaia
Input Options Constitutive Model:	Linear Elastic 🔻	Action: Modify	Analysis Action: Analyze
Property Name	Value	Object: Isotropic	Object: Entire Model
Elastic Modulus =	200000. C		Method: Full Run
Poisson Ratio =	0.28999999		
Shear Modulus =		Existing Materials	Code: MSC.Nastran
Density =		aluminum	Type: Structural
Thermal Expan. Coeff =		b	
Structural Damping Coeff =			Available Jobs
Reference Temperature =			ex_1
emperature Dep/Model Variable F	ïelds:		X
			Job Name
٢	\$		Lob Descripti
		Filter *	
urrent Constitutive Models:			
inear Elastic - [,,,,] - [Active]		New Material Name	
<	3	ring mat	SUBTITLE
			LABEL
ОК	Clear Cancel		
		Description	
Results			Translation Parameters
Action: Create			Solution Type
Object: Quick Plot			Direct Text Input
			Select Superelements
		Input Properties	Subcases
Select Result Cases		Change Material Status	Subcase Select
Default, A1:Static Subcase	> i		
		Apply	Apply

Apply

е

LIST OF REQUIRED PLOTS (total number of plots = <u>10</u>):



Use this reference to verify the results of both analyses.

2) VERTICAL TRANSLATIONAL DISPLACEMENT IN Y DIRECTION (2 plots)

1) FE MODEL WITH LOADS AND BOUNDARY CONDITIONS (1 plot)

- for 1st and 2nd analysis separately



3) AXIAL STRESS "sigma_z" in cylindrical CS (2 plots)

- for 1st and 2nd analysis separately



4) CONSTRAINT FORCES, TRANSLATIONAL, COMPONENT ZZ PLOT (2 plots)

- for 1st and 2nd analysis separately

5) YX GRAPH OF AXIAL STRESS "sigma_z" on top and bottom layer in cylindrical CS (2 plots)

- for 1st and 2nd analysis separately

