# Elastic analysis of a nacelle modeled using Abaqus and its constituent composite plate homogenized using Gmsh4SC

# Elastic analysis of a nacelle modeled using Abaqus and its constituent composite plate homogenized using Gmsh4SC

This tutorial shows how to calculate cross-sectional properties of a composite plate with elastic matrix and unidirectional transversely isotropic elastic fibers for given orientation and material properties using Gmsh4SC. This properties are used to model a nacelle of a wind-turbine in Abaqus, as a homogeneous solid body. Hence the nacelle component can be subjected to structural analysis, with reduced computational effort.

The orientation ie layup of the plates are (0/30/45/90/0/0/90/-45/-30/0) and the materials used are M30S fibers and PMT-F7 epoxy resin with the material properties given below.

The material properties of the PMT-F7 epoxy are, E = 2990 MPa and ? = 0.36.

Properties	M30S Fiber
Longitudinal Stiffness, E1 (MPa)	294,000
Transverse Stiffness, $E_2$ (MPa)	29148
Shear Stiffness, $G_{12}$ (MPa)	11310
In-plane Shear Stiffness, $G_{23}$ (MPa)	10000
Poisson's Ratio, v <sub>12</sub> =v <sub>13</sub>	0.2
Poisson's Ratio, v23	0.46

Fiber properties of

#### M30S

Solution The linear elastic MSG-based modeling of plate is solved in three steps. 1) Elastic homogenization of fiber and matrix at microscale, using Gmsh4SC. 2) Elastic homogenization of the layup (stack of oriented plates), using Gmsh4SC. 3) Modeling of the Nacelle component in Abaqus as a homogeneous body.

1)Elastic homogenization of fiber and matrix at microscale, using Gmsh4SC. Gmsh4SC is available at cdmHUB.org. Go to <u>https://cdmhub.org/tools/scstandard</u> and launch

#### the Gmsh4SC tool.

😜 cdmHUB - The Composites HU 🗙 🕂				- 0	×
	tps:// <b>cdmhub.org</b> /tools/scstandard	F	⊠ ☆	\ ⊡ €	) ≡
					^
Home > Tools > SwiftComp > About	t				
SwiftComp		Launch Tool		819 users, detailed	
By <u>Wenbin Yu<sup>1</sup>, Xin Liu<sup>1</sup></u> 1. Purdue University		Version 1.6.5 - published on 01 Jun 2018 Open source: license   downloa	this	age 0 review(s) (Review s)	
SwiftComp with a GUI developed ba structural solver provided by Calculi	sed on Gmsh and a x.	└── View All Supporting Documents → Share: 🖬 💟 🕄			
About Usage Reviews	Versions Supporting Docs		Re	ecommendations	
Category	Published on		Po	wered by	
Tools	01 Jun 2018		_		~

#### Gmsh4SC

#### 1.1 Creating a new model

Go to File select New and save the model as shown in the figures below.

CdmHUB - The Composites HU X +		- 0	×
$( \rightarrow C )$	··· 🖂 💠		=
			-
<ul> <li>← → C</li></ul>			
	ncel		v
\$			>

## Saving the new model

© cdmHUB - The Composites HU × +	-	ð	×
$(\leftarrow \rightarrow \mathbb{C} \ \textcircled{a}$ (1) $\textcircled{b} \ \textcircled{b}$ https://cdmhub.org/tools/scstandard/session?sess=29102 $\bigtriangledown \ \textcircled{c}$	lii\ 🗉	0	≡
Vp SwiftComp - unt le Teols Window Material Mew Shows All Files (*) Periodicity Mesh SwiftComp Periodicity Mesh SwiftComp Periodicity Mesh SwiftComp Periodicity Mesh Mesoala Calculx Write INP file Import structural Results Imput control Reload Edit file Preview Show hidden files Filename: // fome/sessions/dhananjay/29102/Micro_Plate[geo K // Cancel			
<			>

Saving the new model

1.2 Assigning the material properties:

Go to Material select Thermoelastic and in the pop up window choose the material type (Isotropic, Orthotropic or Anisotropic) and set the material constants and material name, then select Add. Repeat this for all the materials required for the model and close the material properties window. For this tutorial we add the isotropic matrix and transversely isotropic elastic fiber properties under appropriate names.



Assigning matrix material properties

¢	) > c	° 0 🔒	https://cdmhub.o	rg/tools/scstandard	d/session?sess=29	102	⊠ ☆	]	111\	•	≡
E	Upl fi Up Popu	Contractorial propertion     Contractorial     SwiftComp - Micro_Plate     File Iools Window Help     Madules     Material     Thermoelastic     Conduction     Geometry     Common SG     10 S	A Material Pro	perties popic Anisotropic 11D number 29148 E2 11310 G13 0.2 v13 0 Ti 0 alpha22	FIBER 29148 E3 10000 G23 0.46 v23 0 alpha33	Material Name	U 12		Add		Exit
<		- 115									> ×

Assigning fiber material properties

#### 1.3 Creating geometry

Go to Geometry select 2D SG and choose Other 2D SGs and in the pop up window, set the Type of models as Square Pack Microstructure. Modify the Volume fraction of the fiber accordingly and set the Fiber material and Matrix material as required and Add. Close the Window.



#### Creating geometry



#### Creating geometry

#### 1.4 Setting the mesh

Go to Mesh select Generate 2D mesh and choose Generate. To get a finer mesh, go to Input control, click on Edit file and in the pop up window and scroll down to the three Transfinite line commands and increase their value by the factor you wish to increase your mesh. In this

example the mesh is increased by a factor of three. Then save the changes in the window and exit it. Generate the mesh again.



#### Meshing the model

Periodicity     Mesh     Mesh control     Generate 10 mesh	Image: Save     Imag
Generate 2D mesh Refine lines(opt)) Generate Generate 3D mesh Size fields Import SC file SwiftComp Homogenization Dehomogenization Geluix Write INP file Import structural properti- Define step Run Results Input control Reload Edit file Edit file Sort Y Z C 1:1S M < > > Done meshing order	<pre>Micro_Plate.geo % Physical Surface(2) = {82, 86, 88, 84, 60, 62, 64, 66, 68, 70, 72, 74}; Physical Surface(1) = {44, 46, 48, 50, 52, 54, 56, 58}; Transfinite Line { 9, 10, 3, 4, 5, 6, 7, 8, 35, 36, 37, 38, 39, 40, 41, 42, 18, 11, 12, 13, 14, 15, 16, 17, 78, 77, 79, 80} = 3 Using Progression 1; Transfinite Line {33, 19, 21, 23, 25, 27, 29, 31} = 4 Using Progression 1; Transfinite Line {20, 22, 24, 26, 28, 30, 32, 34} = 3 Using Progression 1; Transfinite Surface "*"; Recombine Surface "*"; Mesh.ElementOrder = 2; Mesh.Points = 1; Mesh.SurfaceEdges=1; Mesh.VolumeEdges=1; Mesh.VolumeEdges=1; Mesh.ColorCarouse1=2;</pre>
	C ▼ Tab Width: 8 ▼ Ln 1. Col 1 INS

Meshing the model



#### Meshing the model



#### Meshing the model

#### 1.5 Homogenization

Go to <u>SwiftComp</u> and select Homogenization, and in the window keep the preset options, save the model and run it. The homogenized results will be displayed in a new pop up window.



#### Homogenization

Cumilion - me com	posites no				<i>.</i>	~
↔ → ♂ ☆		Image: The second se	⊘ ☆	lii\ 🖸	0	≡
mbols Documents	Micro_P	late.msh.k 🕱				^
home/s/29102	2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 19 20 21 22 23 24 25 26 27 28 29 30 31 32	1.8581537E+05 4.7178186E+03 4.7178186E+03 1.0123474E-12 4.7178186E+03 1.3410770E+04 5.3580029E+03 1.2852684E-11 4.7178186E+03 5.3580029E+03 1.3410770E+04 -8.1645283E-12 1.0123474E-12 1.2852684E+11 -8.1645283E-12 2.6715452E+03 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 The Effective Compliance Matrix 5.4512675E-06 -1.3702596E-06 -1.3702596E-06 3.3891037E-22 -1.3702596E-06 8.9074889E-05 -3.5106029E-05 -5.3530385E-19 -1.3702596E-06 8.9074889E-05 8.9074889E-05 4.4163518E-19 3.3891037E-22 -5.3530385E-19 4.4163518E-19 3.7431521E-04 0.0000000E+00 0.000000E+00 0.000000E+00 0.000000E+00 0.0000000E+00 0.000000E+00 0.000000E+00 0.0000000E+00 0.000000E+00 0.000000E+00 0.0000000E+00 0.000000E+00 0.000000E+00 The Engineering Constants (Approximated as Orthotropic) E1 = 1.8344358E+05 E2 = 1.1226509E+04 E3 = 1.1226509E+04 E3 = 1.1226509E+04 E3 = 1.1226509E+04 E1 = 2.5136532E-01 nu13= 2.5136532E-01 nu13= 2.5136532E-01 nu23= 3.9411813E-01 Effective Density = 0.000000E+00	0.0000000E+00 0.000000E+00 0.0000000E+00 3.5464058E+03 1.8618912E-12 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 2.8197563E-04 -1.4803945E-19	0.0000000E+00 0.0000000E+00 0.0000000E+00 1.8818912E-12 3.5464058E+03 0.0000000E+00 0.0000000E+00 0.0000000E+00 0.0000000E+00 -1.4803945E-19 2.8197563E-04		•

**Microscale Results** 

1.6 Save the results using an fttp app or note the for the next step.

2) Elastic homogenization of the layup (stack of oriented plates), using Gmsh4SC.

2.1 Open a new model

Go to File select New and save the model as shown previously.

2.2 Assigning the material properties:

Go to Material select Thermoelastic and in the pop up window choose the material type. Use the approximated Orthotropic engineering constants or use the stiffness matrix (more accurate) from the microscale analysis to enter the material properties, then select Add.

🍥 cam+UB - The Composites HUB - Resources: Tools: SwiftComp: Session: 24537 "SwiftComp" — Mozilla Firefox	- 0	×
😻 cdmHUB - The Composite HU X 👔 cdmHUB - The Composite HU X 👔 Group Prof. Yu's Research Gro. X 🚯 Group Prof. Yu's Research Gro. X +		
← → ♂ @ @ O A https://domhub.org/1001/scstandard/session?sess=34857	140% 🖒 🖂 🛛	=
QA Most Visited Generating Started	🗅 Other Bookm	narks
About Explore Community Resources Help	go Logged in (dhananjay)	Î
Home > Resources > Tools > SwiftComp > Session: 34857 "SwiftComp"		
Upple       SwiftComp - Plate.geo         Ite Tools Window Help         Ite Tools Window Help         Ite Tools Window Help         Ite Tools Window Help         Ite Material         Ite Tools Window Help         Ite Tools Window Help <t< td=""><td>27% Bin A D B B d d <sup>351</sup>PM</td><td></td></t<>	27% Bin A D B B d d <sup>351</sup> PM	
🖬 $ ightarrow$ lype nere to search V 🛱 💟 🦷 🔛 💆 💟	2/°C Rain ∧ Ο Ο Ο Μ (40) 8/19/2021	20

Assigning matrix material properties

#### 2.3 Creating geometry

Go to Geometry select 1D SG, choose fast generate and enter the layup as (0/30/45/90/0/90/-45/30/0), but use the square bracket to enter the layups.



#### Creating plate SG geometry



Creating plate SG geometry



Creating layup SG geometry

### 2.4 Setting the mesh

Go to Mesh select Generate 1D mesh and choose Generate.



Creating layup SG mesh

#### 2.5 Homogenization

Go to <u>SwiftComp</u> and select Homogenization, and in the window keep the preset options, save the model and run it. The homogenized results will be displayed in a new pop up window.



Homogenization of the layup

tob - the composites Hork	<b>N</b> CC	ImHUB - The Composites HUEX	Group: Prof. Yu's Research Grou	Group: Prof. Yu's Resea	rch Grou × • User's Manu	al Template - Gmsh 🗶 🚯 Gro	up: Prof. Yu's Research Grou × +			
C @		C A https://cdmhub.org/to	ools/scstandard/session?sess=3	1857					110% 🖒	⊠ 3
Visited 🛛 🌢 Getting Started										🗋 Other Book
Home > Resou	irces >	Tools > SwiftComp > Ses	sion: 34857 "SwiftComp"							
wiftComp									🗙 Terminate 📑	Keep for later
late.msh.k - /hom	ie/cdm	hub/dhananjay/data/se	ssions/34857 - Geany							_ & X
e <u>E</u> dit <u>S</u> earch ⊻ie	W Docu	iment <u>P</u> roject <u>B</u> uild <u>T</u> ool	s <u>H</u> elp	a 📫 🔿						
) 🗕 📥 🖌 🗽 w Open Sa	ve Sa	ve All Revert Close	Back Forward Con	pile Build Execute	Color Chooser	📥 🔮	a 🦾 🌳 Jump to	🐔 Quit		
nbols Documents	DDDDD	D.msh.k 💥 Plate.msh.k 💥	:							
~/data//34857	1 2	The Effective Stiffne	ss Matrix							*
Plate.msh.k	3 4	5.9828462E+004 2.5640769E+004	2.5640769E+004 5.9828462E+004	2.5640769E+004 2.5640769E+004	0.0000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000			
	5	2.5640769E+004 0.0000000E+000	2.5640769E+004 0.0000000E+000	5.9828462E+004 0.0000000E+000	0.0000000E+000 1.7093846E+004	0.0000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000			
	7	0.0000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000	0.000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000	1.7093846E+004 0.0000000E+000	0.0000000E+000 1.7093846E+004			
	9 10	The Effective Complia	nce Matrix							
	11 12	2.2500225E-005	- 6.7500675E- 006	-6.7500675E-006	0.0000000E+000	0.0000000E+000	0.0000000E+000			
	13 14	-6.7500675E-006	2.2500225E-005 -6.7500675E-006	- 6.7500675E-006 2.2500225E-005	0.0000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000	0.0000000E+000 0.0000000E+000			
I	15	0.0000000E+000 0.0000000E+000	0.000000E+000 0.000000E+000	0.0000000E+000	5.8500585E-005	0.0000000E+000 5.8500585E-005	0.0000000E+000 0.0000000E+000			
	17	0.0000000E+000	0.0000000E+000	0.0000000E+000	0.0000000E+000	0.0000000E+000	5.8500585E-005			
	19	The Engineering Const	ants (Approximated as	Orthotropic)						
1	20	E1 = 4.444400	0E+004							
I	22 23	E2 = 4.444400 E3 = 4.444400	0E+004							
	24 25	G12 = 1.709384 G13 = 1.709384	6E+004 6E+004							
	26 27	G23 = 1.709384 nu12= 3.000000	6E+004 0E-001							
I	28 29	nu13= 3.000000 nu23= 3.000000	0E-001 0E-001							
	30 31									
I	32	Effective Density =	0.000000E+000							
	33									
										_
	4									×

Results of layup homogenization

3) Modeling of the Nacelle component in Abaqus as a homogeneous body.

**# Step 3.1.**Create the 3d part shown below using conventional abaqus methodology. Notice we model the nacelle as a 3D homogeneous part. Ensure the part is partition as shown to add the material properties later. The half section of the part is also shown.



#### Nacelle shell



#### Nacelle shell half section

# Step 3.2.' Enter the material properties for the model. First we need to choose the material

properties from the results of the computed effective elastic properties in the previous part. Then we need to convert the Constitutive relations provided as SwiftComp's results into Abaqus's Constitutive relations. This can be done by switching the 4th and 6th rows for the relation and also switching the 4th and 6th column of the stiffness matrix. The relations are provided below. Within the Materials section of Abaqus CAE, we create a material called "Material-1" and add the corresponding properties. We can also import the homogenized properties from the toolbar if we have downloaded the property file from the previous step..

$$\begin{pmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{pmatrix} = \begin{bmatrix} D_{1111} & D_{1122} & D_{1133} & D_{1112} & D_{1113} & D_{1123} \\ & D_{2222} & D_{2233} & D_{2212} & D_{2213} & D_{2223} \\ & D_{3333} & D_{3312} & D_{3313} & D_{3323} \\ & D_{1212} & D_{1213} & D_{1223} \\ & sym & D_{1313} & D_{1323} \\ & & D_{2323} \end{bmatrix} \begin{pmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{pmatrix}$$

Constitutive relations

Ś

$$\begin{cases} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{13} \\ \sigma_{12} \end{cases} = \begin{bmatrix} c_{11} & c_{12} & c_{13} & c_{14} & c_{15} & c_{16} \\ c_{12} & c_{22} & c_{23} & c_{24} & c_{25} & c_{26} \\ c_{13} & c_{23} & c_{33} & c_{34} & c_{35} & c_{36} \\ c_{14} & c_{24} & c_{34} & c_{44} & c_{45} & c_{46} \\ c_{15} & c_{25} & c_{35} & c_{45} & c_{55} & c_{56} \\ c_{16} & c_{26} & c_{36} & c_{46} & c_{56} & c_{66} \end{bmatrix} \begin{cases} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \varepsilon_{23} \\ \varepsilon_{23} \\ \varepsilon_{13} \\ \varepsilon_{212} \end{cases}$$

Constitutive relations

SwiftComp's

output Constitutive relations converted into Abaqus's input Constitutive relations

Abaqus's

SwiftComp's

Abaque-SwiftComp GUI Version 6 16-0	- Model Database: C\Users\17657\Deskton\A	hagus-SwiftComp. GUI. v2.0.2\Nacelle.cae.0/	iewnort 11			- a x
File Model View Viewport M	laterial Section Profile Composite A	sign Special Feature Tools Plug-ins	Help <b>k</b> ?			_ 8 ×
I D 🚅 🖬 🖷 🛔 🛔 👘 🤊	Property defaults	V 🖬 (Ra 🕇 🗖 🖬 🥿 i 🖩			7111234 大日中 🖉 🔍 🕅	tl : 🖳 📻
						1
Model Results Material Library	Module: Property V Model:	Model-1 V Part: Part-6 V		Import homogenized properties		
Model Database     Model:     Model:	2: III       1: IIII       1: IIII       1: IIIII       1: IIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIIII	♦ In SSE O C Sum Analy Meeter	nport homogenized properties Model Source AE @ SwiftComp homogenized properties file Comp homogenized properties file Socopic model OK OK	X 857/Deiktop/Abaque/SM 2 Cancel	¥ z	
< >>	z	x				²S SIMULIA
508520 elements have be The model database has	en generated on part: Part-6 been saved to 'C:\Users\17657	Desktop\Abaqus-SwiftComp_GUI_	v2.0.2\Nacelle.cae".			^
Application restarted a	fter 165 minutes of idle time;	the license will be checked	v2.u.2\macelle.cae". within the next 3 minutes.			▼
€ P Type here to search	0 #	i 💽 🖬 📾 🧕	S 🖬 💠 🛐 🗉		🌙 21°C Mostly clear \land ট্রি 🖨 📾 🌈 🕼	4:36 AM 8/7/2021

#### Importing Material properties

# Step 3.3.' Create an anisotropic elastic material and add the homogenized material properties as explained in the previous step. Create and assign a section to the part.

<u>File Model View</u> Viewport N	Nat <u>e</u> rial <u>S</u> ection <u>P</u> rofile <u>C</u> omposite	Assign Special Feature Tools Plug-ins He	k?		_ 6 X
i 🗅 💣 🖬 🖶 🛔 i 🚺 🕷	🕽 Property defaults 🗸 📁 🗸 🗼	All 🛛 🖓 🖷 🕼 🗖 🏟 🔛 🗖	〒   回日   自己   ●	1 🗏 🔂 😐 🗄 🗛 🌐 🎒 🕼 🗇 🎁 🗓 x 1	↓□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□□
			000		
	Module: Property Model	Model-1 Part: Dart-6			
Model Results Material Library	Module. Property Mode			1.50.050	
Model Database M 🗘 🗈 🗞 😘					
🗏 🎎 Models (1)	1. 🚍				l Y
B Parts (6)	1 m				
Materials		Talk Manufal			
S Calibrations	- <sup>12</sup> 231	Edit Material			
Sections	🖕 🔆 .	Name: Material-1			Z X
Assembly	🕂 🌐	Description:			
⊞o⊊ Steps (1)					
Field Output Requests		Material Behaviors			
History Output Requests				Sector and the sector of the s	
Le ALE Adaptive Mesh Constra	- <u>11 1-</u>				
1 Interactions					A. C.
뮬 Interaction Properties	<u> </u>				S.(
Contact Controls	+ /	General Mechanical Them	I <u>E</u> lectrical/Magnetic <u>O</u> ther		
Contact Intializations	<u></u>	Elasticity	► <u>E</u> lastic		
Constraints	-15 11.	Plasticity	► <u>Hyperelastic</u>		
Connector Sections		Damage for Duct	2 Metals Hyperfoam		
B J Fields	Han Hay,	Damage for Fiber	Reinforced Composites  Hypoelastic		
Loads	(XYZ)	Damage for Elast	mers Porous Elastic	1	
- 🛄 BCs	24	De <u>f</u> ormation Plas	city <u>V</u> iscoelastic		
- E Predefined Fields	····.	Damping			
Remeshing Rules		Expansion			
Annotations		Ens			
🗄 🛊 🛱 Analysis		Viscosity			
- 🛃 Jobs		Super Elasticity			
Adaptivity Processes					
Co-executions					
an optimization receives	v				
	T				
	Z	Х	Cancel		
					2
< 2					j∕S simulia
The instance Part-3-2	vas translated by 0., 0., 1.	605E+03 with respect to the assembly con	coordinate system		^
The instance Part-3-2	vas translated by 0., 75., 0	). with respect to the assembly coord	nate system		
warning: Cannot contin	ie yetcomplete the step of	cancel the procedure.			10:02 AA4
Type here to search	0	🗏 🖸 🦰 🛄	🖷 🌩 🖳		21°C AQI 36 ^ B C K 40 8/7/2021

Create an anisotropic elastic material



#### Create a section



#### assign the section

' # Step 3.4. Now we assign the material orientation for the part. Go to Assign material orientation -> select the end sections of the part and -> Done -> Select a CSYS (use default

orientation or other method) -> set the global orientation to both as shown. They will have opposite orientation if the plies are symmetric. But we are not using symetric model for the nacelle. -> Done –



#### Assign material orientation

(Image(N3.42.png, desc="Assign material orientation") failed - File not found)

*\* # Step 3.5.* Select the top and bottom sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the Coordinate system orientation to both as shown.
 -> Define -> addition axis of rotation -> axis 2 as 90 degree -> Continue -> OK -> Done -



#### Assign material orientation



*# Step 3.6.* Select the side sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the Coordinate system orientation to both as shown. -> Define -> addition axis of rotation -> axis 2 as 90 degree -> Continue -> OK . -> Done --





# Step 3.7. Select the top right curved sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the discrete orientation as shown. -> Define -> Primary axis orientation -> choose edge and flip direction if needed to make the axis point towards a clockwise direction -> Choose the surfaces for the normal axis definition -> Continue -> OK -> Done -









*# Step 3.8.* Select the bottom right curved sections of the part and -> Done -> Select a CSYS (use default orientation or other method) -> set the discrete orientation as shown. -> Define -> Primary axis orientation -> choose edge and flip direction if needed to make the axis point towards a clockwise direction -> Choose the surfaces for the normal axis definition -> Continue -> OK -> Done -







#### # Step 3.9. The final orientation should look like this



Assign material orientation

# Step 3.10. Mesh the part and continue with the traditional structural analysis with reduced mesh elements.

