Predictions of local/global stress/strain fields in composite structures

In this problem, we will try to show how to analyze the local-global fields in composite structures using <u>SwiftComp</u>-Abaqus-GUI.

The figure below can summarize how to do the local global analysis.



Let the material properties a fiber (T300) property be: : E_{11} =230 GPA, E_{22} =15 GPA, v_{12} =0.20, v23=_{0.0714}, G_{12} =15GPa, G_7 =3.928GPa. and matrix (3501-6 epoxy) be: E =4.2GP, ?=0.34 The composite lay-up:

[0/90/45],,s,,

Thickness of each ply=0.00025m

"Soden, P. D., Hinton M. J. and Kaddour, A. S., Lamina properties, lay-up configurations and loading conditions for a range of fibre reinforced composite laminates. Compos. Sci. Technol., 1998, 58(7), 1011"

Major steps to perform local-global analysis

Step 1: Input material properties

ne.	Fiber				
crip	tion:				
1ater	rial Behaviors				
ensit	ty				
astic	5				
<u>3</u> ene	eral <u>M</u> echanical	<u>T</u> hermal <u>E</u> lect	trical/Magnetic <u>O</u> t	her	
asuu	·				
ype:	Engineering Cor	nstants 🗸			 Suboption
] Us	e temperature-de	pendent data			
_					
lumk	her of field variabl	C.71 W W			
lumb	ber of field variabl				
lumt 1odu	ber of field variabl uli time scale (for v	viscoelasticity): Lo	ng-term 🖌		
lumb 1odu] No	ber of field variabl uli time scale (for v o compression	viscoelasticity): Lo	ng-term 🖌		
lumb 1odu] No] No	ber of field variabl uli time scale (for v o compression o tension	viscoelasticity): Lo	ng-term 🔽		
lumb 1odu] No] No Data	ber of field variabl uli time scale (for v o compression o tension a	viscoelasticity): Lo	ng-term 🔽		
lumb 1odu] No] No Data	ber of field variabl uli time scale (for v compression c tension a E1	viscoelasticity): Lo	ng-term	Nu12	Nu13
lumb 1odu] No] No Data	ber of field variabl uli time scale (for v o compression o tension a E1 230000000000	viscoelasticity): Lo E2 1500000000	E3 1500000000	Nu12 0.2	Nu13 0.2

		×
		1
l <u>T</u> hermal <u>E</u> lectrical/	'Magnetic <u>O</u> ther	✓
		▼ Suboptions
ependent data		
les: 0		
viscoelasticity): Long-te	urm V	
inconstantly, Long a		
Poisson's Ratio		
0.34		
	I Thermal Electrical/ ependent data ependent ependent ependent ependent ependent ependent ependent ependent ependent ependent ependent ependent ependent ependent ependent e	I Thermal Electrical/Magnetic Other ependent data les: O viscoelasticity): Long-term Poisson's Ratio 0.34

Step 2: Select appropriate SG

a. Select 3D SG that represent the current example

 b. 3D SG wizard shows up c. Select spherical inclusion as microstructure d. Add inclusion volume fraction e. Select material properties for inclusion and matrix f. Click on OK to generate the SG g. See generated 2D SG 2D Structure Genome - Unit Cell 	×
Select a profile Sequare Hexagonal Geometry Interphase Fiber Volume fraction (vf_f) Radius (r) O 0.60 O Note: 0 < vf_f + vf_i <= 0.90	
OK f Apply Cancel	

(Image(Problem-4bb.JPG) failed - File not found) Step 3- Homogenization- 3D effective properties



- b. Homogenization wizard shows up (see below)
- c. Select 3D (solid) Model
- d. Select analysis type, elastic
- e. Click on OK to start homogenization
- f. See the predicted 3D effective properties
- 💠 Homogenization

Model source				
CAE O Input file				
Model: Model-1 🖌 Part	: Laminate 🖌		h	
Macroscopic model			N	
Dimension Dimen	sionally reducible structures			
O 1D (Beam) Specific	model: Classical	~		
🔿 2D (Shell)				
□ Omega: Note: Provide omega if th	e part is not a line, rectangle or cu	be		
Omega: Note: Provide omega if th Options	e part is not a line, rectangle or cu	be		
Omega: Note: Provide omega if th Options Analysis type:	e part is not a line, rectangle or cul Elastic	be		
Omega: Omega: Note: Provide omega if th Options Analysis type: Element type:	e part is not a line, rectangle or cu Elastic Regular	be		
Omega: Omega: Note: Provide omega if th Options Analysis type: Element type: Elemental orientation:	e part is not a line, rectangle or cul Elastic Regular Global	be d		
Omega: Note: Provide omega if th Options Analysis type: Element type: Elemental orientation: Temperature distribution:	e part is not a line, rectangle or cul Elastic Regular Global Uniform	be d		
□ Omega: Note: Provide omega if th Options Analysis type: Element type: Elemental orientation: Temperature distribution: Aperiodic □ y1 □ y2 □ y3	e part is not a line, rectangle or cul Elastic Regular Global Uniform	be d		
 □ Omega: Note: Provide omega if th Options Analysis type: Element type: Elemental orientation: Temperature distribution: Aperiodic □ y1 □ y2 □ y3 □ Only generate input file. 	e part is not a line, rectangle or cul Elastic Regular Global Uniform Do not run SwiftComp.	be De De De De De De De De De De De De De		

Step 4: Export predicted effective properties to create a new model a. Export the predicted effective properties



b. A new model is automatically generated, this is to be used for generating a global model c. Predicted effective properties exported as engineering constants



d. Predicted effective properties exported as engineering constants

Step 5: Generate the global model

- a. Click on Part and name as Part-1
- b. Select 'Shell' from from Shape and 'Planer' from type
- c. A new model is generated as shown (0.2m and thickness=0.15m)

1	💠 Create Part	Х	
66	Name: Part-1		
	Modeling Space		
1	🧿 3D 🔿 2D Planar	⊖ Axisymmetri∂	
li, It,	Туре	Options	
2	Deformable Discrete rigid		
-+	O Analytical rigid	None available	
L	🔿 Eulerian		
(XYZ)	Base Feature		
51 I	Shape Type		
//>	O Solid Plana		
	Shell Extrus Bevolu	ion	
C.K	O Wire Sweep	b	
🏠 🏠	O Point		
	Approximate size: 200		
	Continue C	Cancel	



Step 6: Create and Assign Section'

- a. Click on Section and name as 'Composite'
- b. Select 'Shell' from from Category and 'Composite' from type
- c. Edit section wizard shows up
- d. Add section properties as shown and click OK

Name: Composite Category Type O Solid Homogeneous Shell Composite O Beam Internorane				
Category Type Solid Homogeneous Shell Composite				
Solid Homogeneous Shell Composite Beam Memorane				
Shell Composite Beam Memorane				
Shell Composite Shell Iviembrane	_			
() Beam				
Surface	h			
O Fluid General Shell Stiff	ness			
○ Other				
Continue Ca	ncel			
💠 Edit Section				
Name: CompositeA				
Type: Shell / Continuum Shell,	Composite			
Section integration: During a		Refore analysis		
Layup name:				
Basic Advanced				
Thickness integration rule: 🔘	Simpson 🔿	Gauss		
Symmetric layers				
Material T	hickness	Orientation Angle	Integration Points	Ply Name
	0.00025	n	3	Δ
nexP2_nSG2_3D_S4pbc_engi		v		<u> </u>
nexP2_nSG2_3D_S4pbc_engi nexP2_nSG2_3D_S4pbc_engi	0.00025	90	3	В
nexP2_nSG2_3D_S4pbc_engi nexP2_nSG2_3D_S4pbc_engi nexP2_nSG2_3D_S4pbc_engi	0.00025	90 45	3 3	B
hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi	0.00025 0.00025 0.00025	90 45 45	3 3 3	d D
hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi hexP2_nSG2_3D_S4pbc_engi	0.00025 0.00025 0.00025 0.00025	90 45 45 90	3 3 3 3	B C D E

e. Assign material orientation

f. Click on Axis 3



Step 7: Create Assembly and Steps '

- a. Select Assembly
- b. Select 'Part-1' from Create Instance
- c. Click OK

(Image(Problem-4M7.jpg) failed - File not found)

e. Create steps, accept the default setting

-			e		
	Nan	Manager	Procedure	Nigeom	Time
	/ Initi	al	(Initial)	N/A	N/A
	/ Step	p-1	Static, General	OFF	1

Step 8: Create load and boundary conditions '

- a. Select load from Module
- b. Select load
- c. Name the load as 'Load-1'



- f. Select the area to be loaded
- g. Add the load
- youtube link:
- Step 9: Create boundary conditions '
- a. Select load from Module
- b. Boundary Conditions (BC)
- c. Name the BC as 'BC-1' and Select 'Step-1'
- d. Select 'Mechanical' from Category and Symmetry type BC
- e. Click on OK



f. Select the edge for BC

g. Select ENCASTER



Step 10: Create Mesh '

Step 11: Run the analysis '

Step 11: Results of the analysis '

~	s	🛛 Mises 🛛 🔦 🗄 🕂 🥐 🔍 🔩 🔯 輝 🗄 🛔 🛔 🖡 🖡 🖓 🗚 🔤 🖓 🖓 🖓 🏣 🌆 🎼 🏀 🖓 🖓 🖓 🖓 🖓 🖓 🖓 🖓 🖓 🖓 🖓 🖓 🖓
	Module:	Visualization V Model: 定 C:/Users/hsertse/Desktop/abaqus-swiftcomp_gui/Test1.odb V
ð.		S, Mises SNEG, (fraction = -1.0), Layer = 1 (Avg: 75%)
		+7.285e+09 +6.678e+09 +5.671e+09 +5.464e+09 +4.857e+09 +4.250e+09 +4.250e+09
	R. H	+3.036e+09 +2.429e+09 +1.823e+09
		+1.216e+09 +6.088e+08 +1.896e+06
	1	
	₩	
	A 8	
	<u> </u>	
	K 💼	
	5	

Step 12: Obtain global strains '

- a. Click on probe values to obtain global strain
- b. Click on a point on global structure
- c. Select nodes and all direct from probe values wizard
- d. The following strain values can be obtained, strain e_{11} =-0.042709, e_{22} =0.00212116,
- $2e_{12}$ =0.00788458, and all others are zero

Step 13: Run dehomogenization '

a. Go back to micromechanical analysis with hexagonal SG and click on dehomogenization



- b. Add global strain obtain in step 12 to obtain local field in 0 degree lamina
- C. Click on OK

💠 Dehomogenization	I	×		
SG model source				
● CAE ○ SwiftCon	np Input file			
hexP2_nSG2_3D_S4pb	ć			
Macroscopic analysis	results			
Displacements				
¥1	v 2	v 3		
0.0	0.0	0.0		
Rotations				
1.0	0.0	0.0		
0.0	1.0	0.0		
0.0	0.0	1.0		
Generalized strains				
ensilon 11	ensilon22	encilon 33		
-0.042709	0.003132	0.0		
2ensilon23	2epsilop13	2ensilon12		
0.0	0.0	0.0078845		
Additional inputs				
temperature increme	nt 0			
		Connact		
OK				

Step 14: Create view port and show both global and micromechanical local field analysis '

