复合材料 Abaqus 仿真分析——精讲版

本文以一个非常简单的复合材料层合板为例,应用 Abaqus/CAE对其进行线性静态分析。一块边 长为 254mm的方形两层层合板,两层厚度均为 2.54mm,第一层铺层角 45°,第二层铺层角-45°; 板的四边完全固支,板的上表面受到 689.4kpa 的压强。各单层的材料相同,材料属性如下: El=276GPa E2=6.9GPa, E3=5.2GPa, γ12=0.25, G12=3.4GPa, G13=3.4GPa, G23=3.4G。

定义模型的几何形状

创建一个具有平面壳体单元基本特征的三维变形体,在草图环境绘制板的几何形状如下图:

Create Par	t	×					
Name: Part-1							
Modeling Space							
SD 2D Planar Axisymmetric							
Туре		Options					
 Deformal Discrete r Analytica Eulerian 	None available						
Base Feature	e						
Shape	Туре						
Solid	Planar						
Shell	Extrusio	on					
Wire	Revolu	tion					
Point	Sweep						
ر <u></u> ر		00					
Approximate	size: 20	JU					
Continue		Cancel					

	*
254.	

定义材料属性和局部材料方向

Edit Material						×
Name: Material-la	aminate					
Description:						Edit
Material Behavio	ors					
Elastic						
General Mech	anical Thermal	Other				Delete
<u>G</u> eneral <u>M</u> ech		<u>o</u> tnei				Delete
Elastic						
Type: Lamina						 Suboptions
Use temperat	ure-dependent d	ata				
Number of field	variables:	0 🚔				
Moduli time sca	e (for viscoelasti	city): Long-tern	י י			
No compressi	on					
No tension						
Data						
El 1 276000	6900	0.25	G12 3400	G13 3400	G23 3400	
		0.20		2.00		
	OK				Cancel	

Create coordinate system



een created. yet--complete the step or cancel the procedure.

定义局部坐标系,对于像本例这样的简单几何体,本可以不用另外建立局部坐标系,但笔者还是 在本例中用了局部坐标系,主要是考虑到以后再复杂问题中会经常用到这一方法。

创建铺层



或者使用菜单栏



💵 Create Composite Layup 🛛 🗾						
Name: CompositeLayup-1						
Initial ply count: 3						
Element Type						
Conventional Shell						
Continuum Shell						
Solid						
Continue Cancel						

注意这三种单元类型的区别,详情请查看

此处使用全局坐标系

								Ł
	🔳 Edi	t Compos	ite Layup					
2 1 2 1 4 4 4	Name	Compos	iteLayup-1					
	Eleme	nt type: C	Conventional She	ell Descr	iption: Lamir	nate with two	plies	
	 _⊂ Layu	p Orientat	tion					
2 1 2 1 2 1 1	Defin	ition: Par	t global		Create	e		
	Pa	rt coordir	ate system					
and the second								
	Sectio	n integrat	ion: () During a	nalvsis 🔘 Befo	ore analysis			
	Thickn	ess integr	ation rule: Si	mpson © Gau	ss			
	Plies	Offset	Shell Paramete	rs Display				
	🗖 🗖 Ma	ke calcula	ated sections syr	nmetric		🖆 🖶	11 II I	🗄 🚌 🗲
		Ply Nan	ne Region	Material	Thickness	CSYS	Rotation Angle	ntegration Points
	1 4	Ply-1	(Picked)	aterial-lamina	2.54	<layup></layup>	45	3
	2 🗸	Ply-2	(Picked)	aterial-lamina	2.54	<layup></layup>	-45	3
r dialog								
.ipboard.								
.ipboard.								

ite Assign Special Feature Tools Plug-ins Help *?	Eleme Layu Defin Pa	t Composite L CompositeLant type: Conve p Orientation ition: Part glo rt coordinate	ayup ayup-1 entional Sho bal system	ell Descr	iption: Lamin	nate with two	o plies	
	Thickn Plies	offset She ke calculated	n rule: Si Il Paramete sections syn Region	mpson © Gau rrs Display mmetric Material	Thickness	te 🚌	ti III	tegratior
	1 v 2 v	Ply-1 Ply-2	(Picked) (Picked)	aterial-lamina aterial-lamina	2.54 2.54	<layup> <layup></layup></layup>	45 -45	3
yup editor dialog the procedure. o the clipboard. o the clipboard.]			Canc	al

	E	dit	Composite	Layup									×
Na	Name: CompositeLayup-1												
Ele	Element type: Conventional Shell Description: Laminate with two plies												
7	a	yup	Orientation										
D)e	fini	tion: Part ol	obal		Creat	e)						
Se Th	Part coordinate system Section integration: O During analysis Before analysis Thickness integration rule: Simpson Gauss												
P		es	Offset Sh	ell Paramete	ers Display				rt-	- <u>*</u> ±	18±	1 =	× I
		Mal	ke calculated	sections sy	mmetric								9 Y
			Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	ntegration Points				
1	L	~	Ply-1	(Picked)	aterial-lamina	2.54	<layup></layup>	45	3				
2	2	~	Ply-2	(Picked)	aterial-lamina	2.54	<layup></layup>	-45	3				

使用用户自定义坐标系

Edi	t Composite I	Layup						×
Name:	Name: CompositeLayup-1							
Eleme	Element type: Conventional Shell Description: Laminate with two plies							
Layup Orientation Definition: Coordinate system Create Datum csys-1 Select Create Normal direction: Axis 1 Axis 2 Axis 3 Additional rotation: None Angle: Distribution: Create Section integration: During analysis Before analysis Thickness integration rule: Simpson Gauss Plies Offset Shell Parameters Display								
Ma	ke calculated	sections sy	mmetric					御殿 御殿 副戦 س
	Ply Name	Region	Material	Thickness	CSYS	Rotation Angle	integration Points	
1 4	Ply-1	(Picked)	aterial-lamina	2 54				
2 🗸	Plv-2			2.51	<layup></layup>	45	3	
Rotation angle depends on the coordinate system defined by user. Par example, if x-axe in the user defined system is parallel to the direction of fiber; we should replace the angles by 0 and 90.						45 -45	3	
		(PICKed)	aterial-lamina R de de W	2.54 otation an efined by efined system e should n	<layup> <layup> ngle depe user. Par stem is pa replace th</layup></layup>	45 -45 ands on the example rallel to the angles here	3 3 he coordin e, if x-axe the directin by 0 and 9	nate system in the user on of fiber; 0.

使用全局坐标系和局部坐标系的区别在下面这一步可以查看

如果使用全局坐标系,会有方向指示,如果使用用户自定义坐标系,在层中没有方向指示

可以通过,工具——查询,来检查铺层(Tool ----Query----ply stack plot)

Case 1 全局坐标系



使用局部坐标系



生成装配件、定义分析步和输出要求

Create Instance					
Parts					
Part-1					
Instance Type					
Dependent (mesh on part)					
Independent (mesh on instance)					
Note: To change a Dependent instance's mesh, you must edit its part's mesh.					
Auto-offset from other instances					
OK Apply Cancel					

定义分析步,保留各项默认值即可。

📑 Create Step	
Name: Step-1	
Insert new step after	
Initial	
Procedure type: General	
Dynamic, Explicit	
Dynamic, Temp-disp, Explicit	
Geostatic	



场输出要求和历史输出要求都按默认的输出方式。

为了结果中能查看每个 ply 的输出情况,应该在场输出中进行一些修改

Model Results	Module: Step	Edit Field Output Request
 Field Output Requests Manager Name Step-1 F-Output-1 Created 		Name: F-Output-1 Step: Step-1 Procedure: Static, General Domain: Whole model Frequency: Every n increments Iming: Output at exact times
Step procedure: Static, General Variables: Preselected defaults Status: Created in this step Create Copy Rename Surfaces Connector Assignments Connector Assignments Connector Assignments Connector Assignments Connector Assignments Connector Assignments Connector Assignments Field Output Requests (1) Connector Assignments Connector Assignments Conference Connector Assignments Conference Conference Contents of viewport "View Contents of viewport "View	created. created. complete the step wport: 1" have bee alayup-1" has been wport: 1" have bee wport: 1" have bee	Output Variables Select from list below Preselected defaults All Edit variables CDISP,CF,CSTRESS,LE,PE,PEEQ,PEMAG,RF,S,U, Stresses Strains Displacement/Velocity/Acceleration Forces/Reactions Contact Forces/Reactions Failure/Fracture Failure/Fracture Use defaults Specify: Use defaults Specify: Include local coordinate directions when available OK Cancel

Model Results Module: Step	Edit Field Output Request
 ■ Field Output Requests Manager Name Step-1 ✓ F-Output-1 Created 	Name: F-Output-1 Step: Step-1 Procedure: Static, General Domain: Composite layup : Part-1-1.CompositeLayup-1 Frequency: Every n increments n: 1 Timing: Output at exact times .
Step procedure: Static, General Variables: Preselected defaults Status: Created in this step Create Copy Rename Delete Create Copy Rename Delete Create Copy Rename Delete Surfaces Connector Assignments Steps (2) Field Output Requests (1) Field Output Requests (1) Field Output Requests (1) A new model database has been created. The model "Model-1" has been created. The model "Model-1" has been created. Marning: Cannot continue yetcomplete the step The contents of viewport "Viewport: 1" have bee The contents of viewport "Viewport: 1" have bee The contents of viewport "Viewport: 1" have bee The contents of viewport "Viewport: 1" have bee	Output Variables Select from list below @ Preselected defaults @ All @ Edit variables LE,PE,PEEQ,PEMAG,S, Stresses Strains Displacement/Velocity/Acceleration Forces/Reactions Energy Failure/Fracture Thermal Output at Section Points @ Selected points for each ply: Top @ Middle Bottom All section points in all plies Specify: Include local coordinate directions when available

(场输出 field output 和历史输出 history output 的区别, refer to user s manual)

规定边界条件和施加载荷 (689.4 KPa and all edges : encastre)

🗉 Edit Load		
Name: Load	d-1	
Type: Pressure		
Step: Step-1 (Static, General)		
Region: (Picked)		
Distribution:	Uniform	Create
Magnitude:	0.6894	
Amplitude:	(Ramp)	Create
Oł	(Cancel

定义完边界条件和载荷后模型会有如下显示



划分网格和定义作业

定义单元类型 S8R5(The differences of all element types should be clear for analysis)

以上内容仅为本文档的试下载部分,为可阅读页数的一半内容。如 要下载或阅读全文,请访问: <u>https://d.book118.com/12801111611</u> 5007002