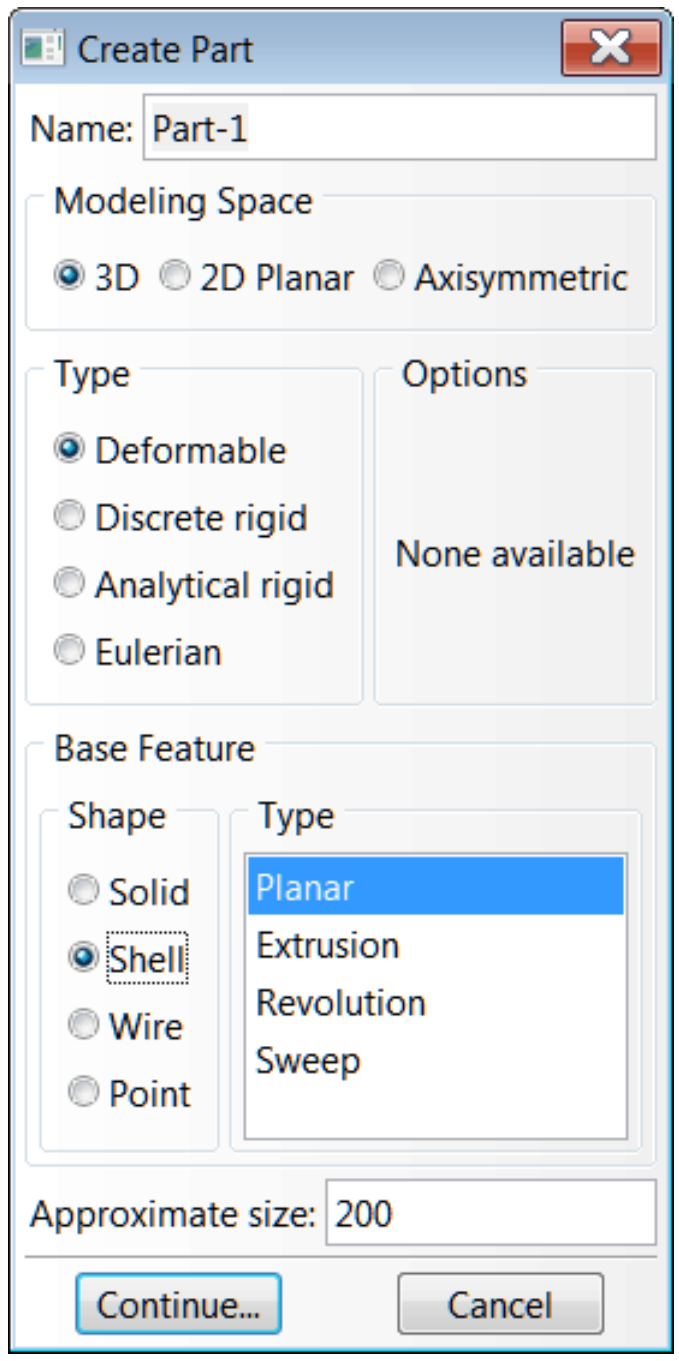
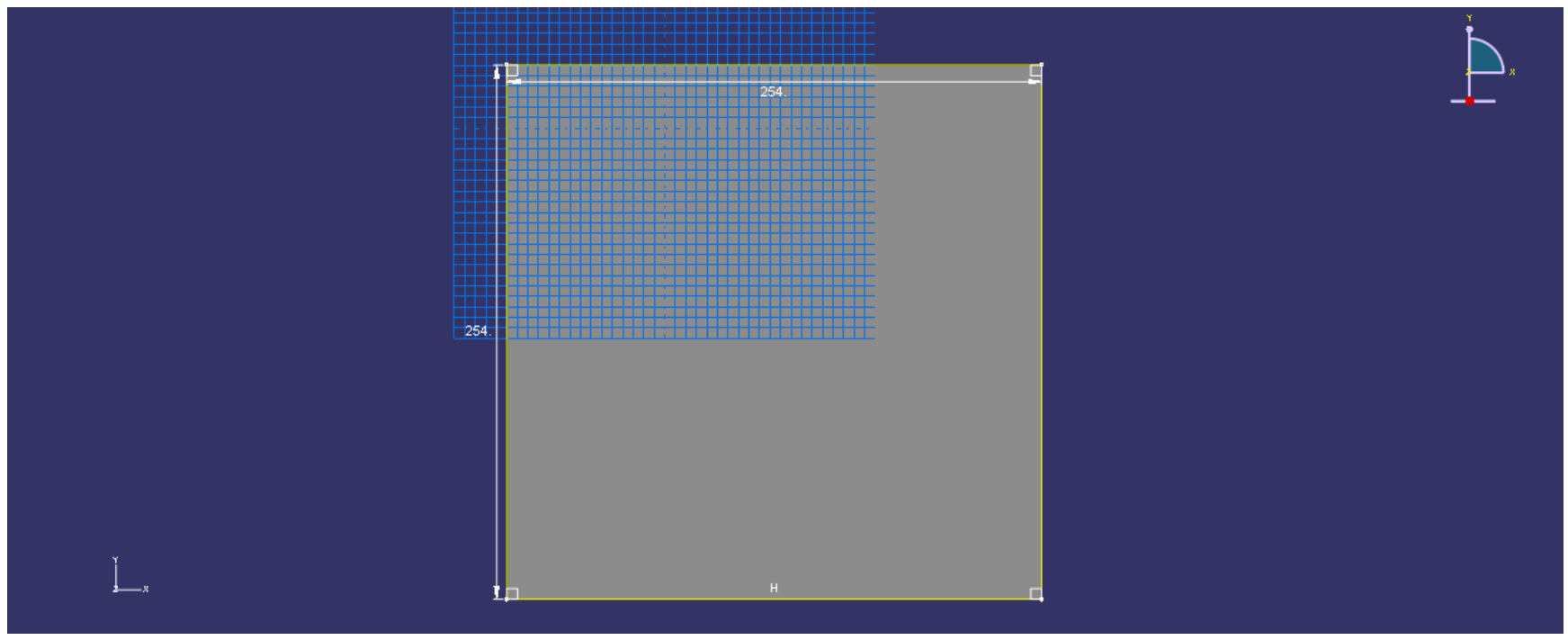
复合材料 Abaqus 仿真分析——精讲版

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：E1=276GPa，E2=6.9GPa，E3=5.2GPa，γ12=0.25，G12=3.4GPa，G13=3.4GPa，G23=3.4G。

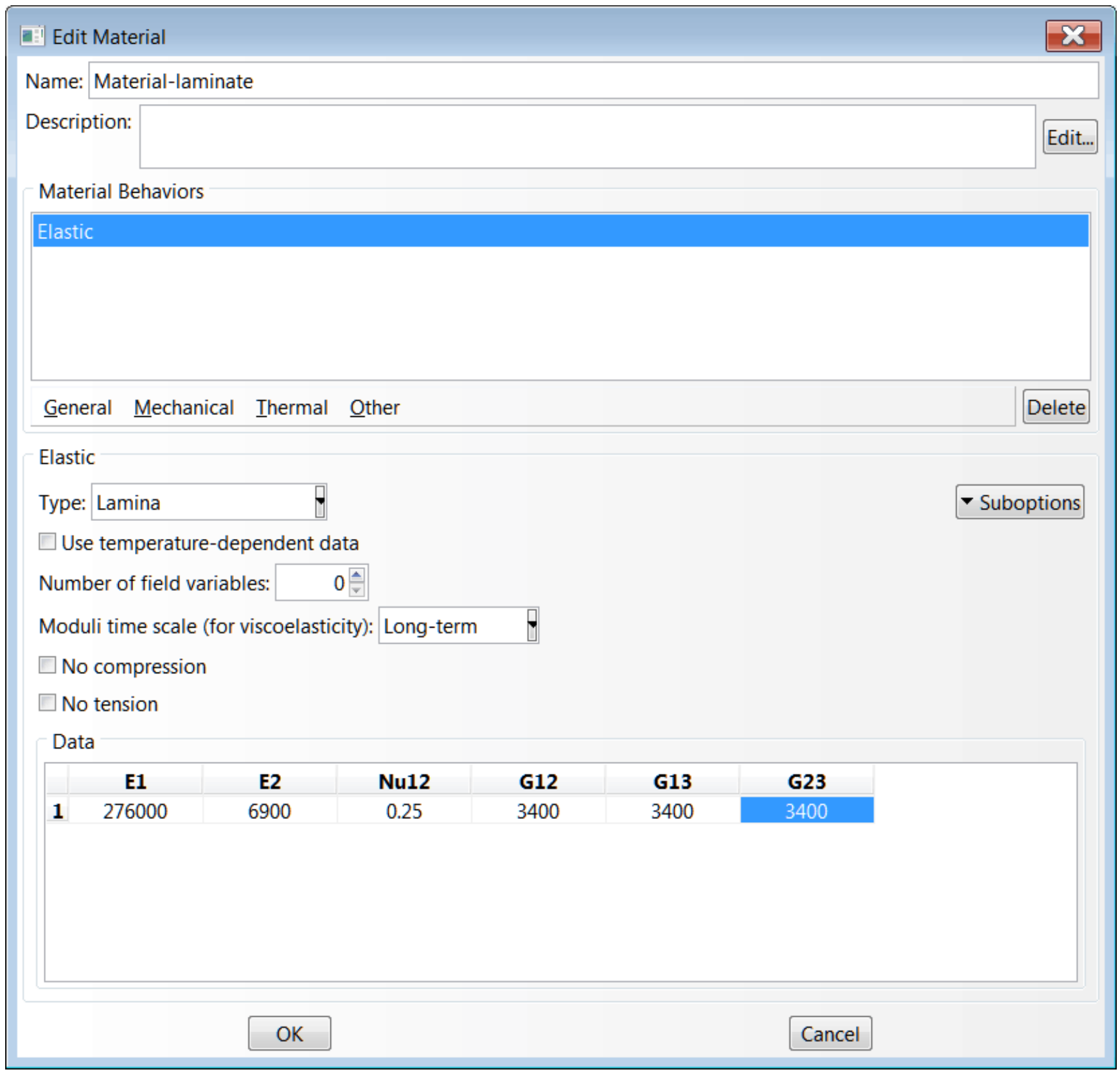
定义模型的几何形状

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：

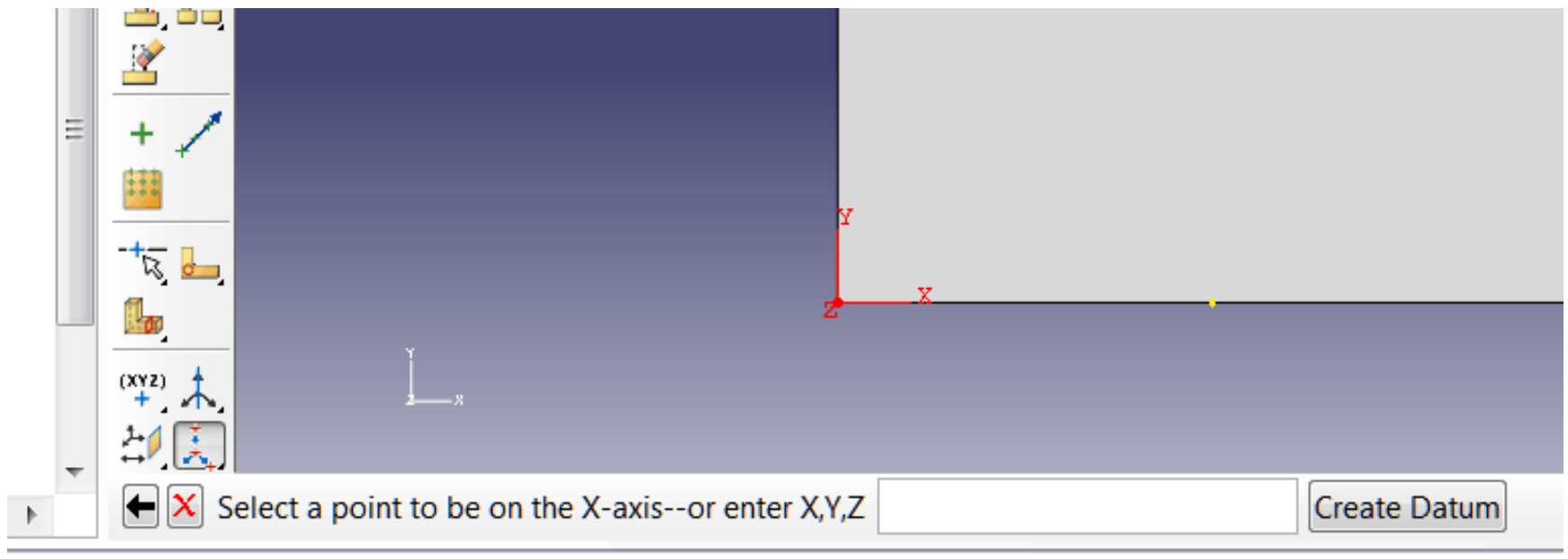




定义材料属性和局部材料方向



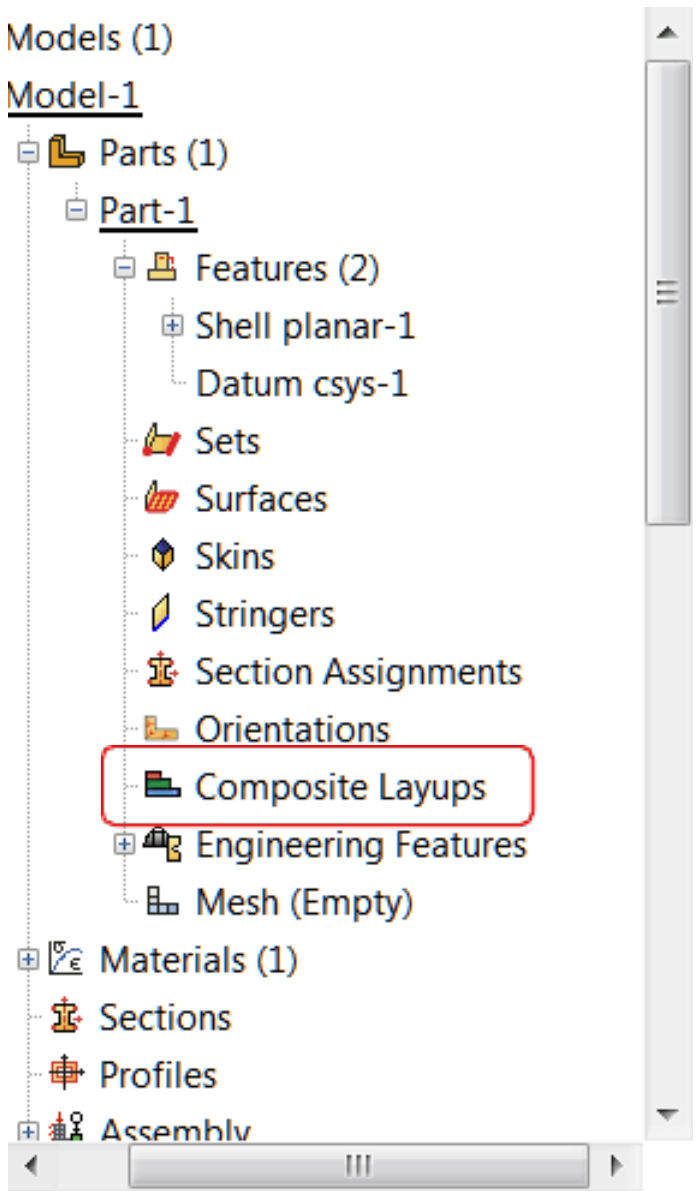
Create coordinate system



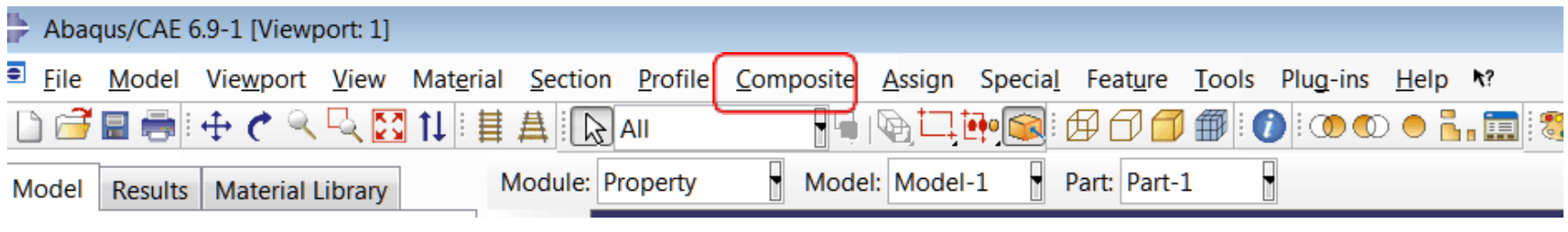
been created.
een created.
yet--complete the step or cancel the procedure.

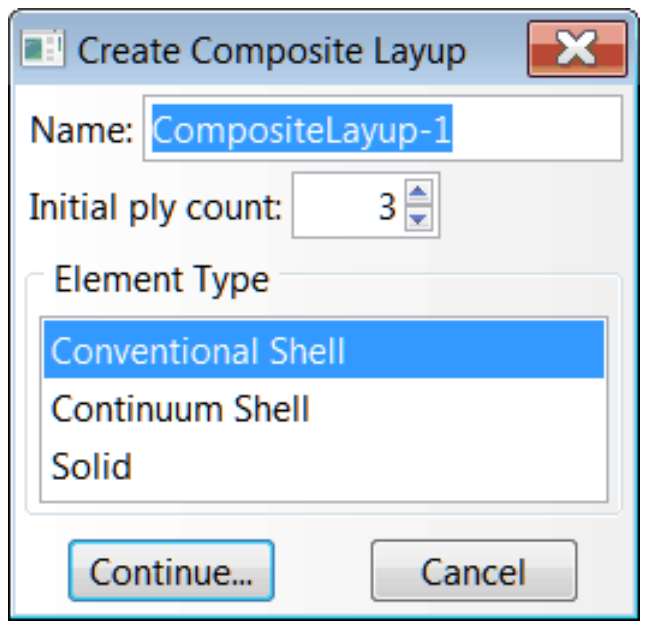
定义局部坐标系，对于像本例这样的简单几何体，本可以不用另外建立局部坐标系，但笔者还是在本例中用了局部坐标系，主要是考虑到以后再复杂问题中会经常用到这一方法。

创建铺层



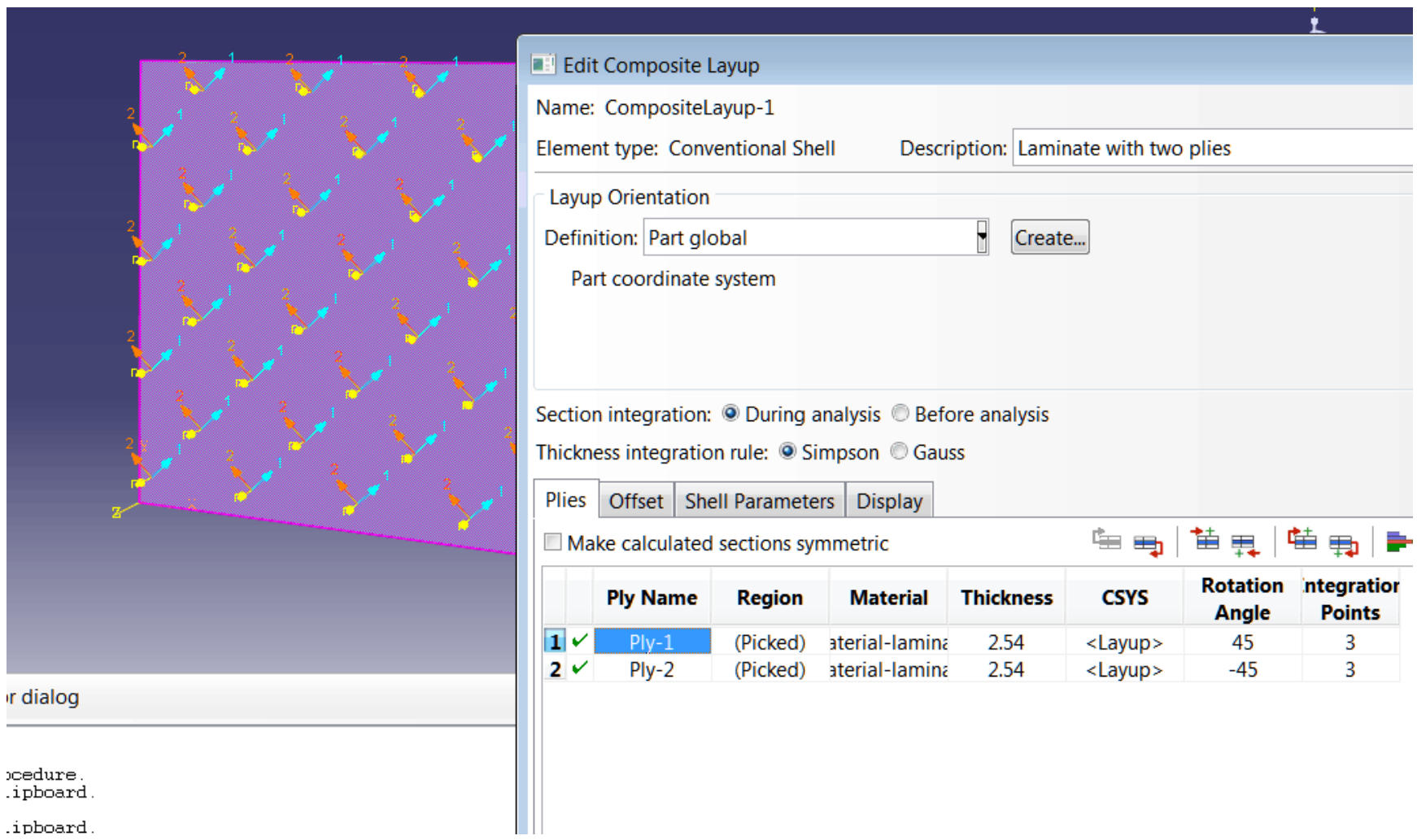
或者使用菜单栏

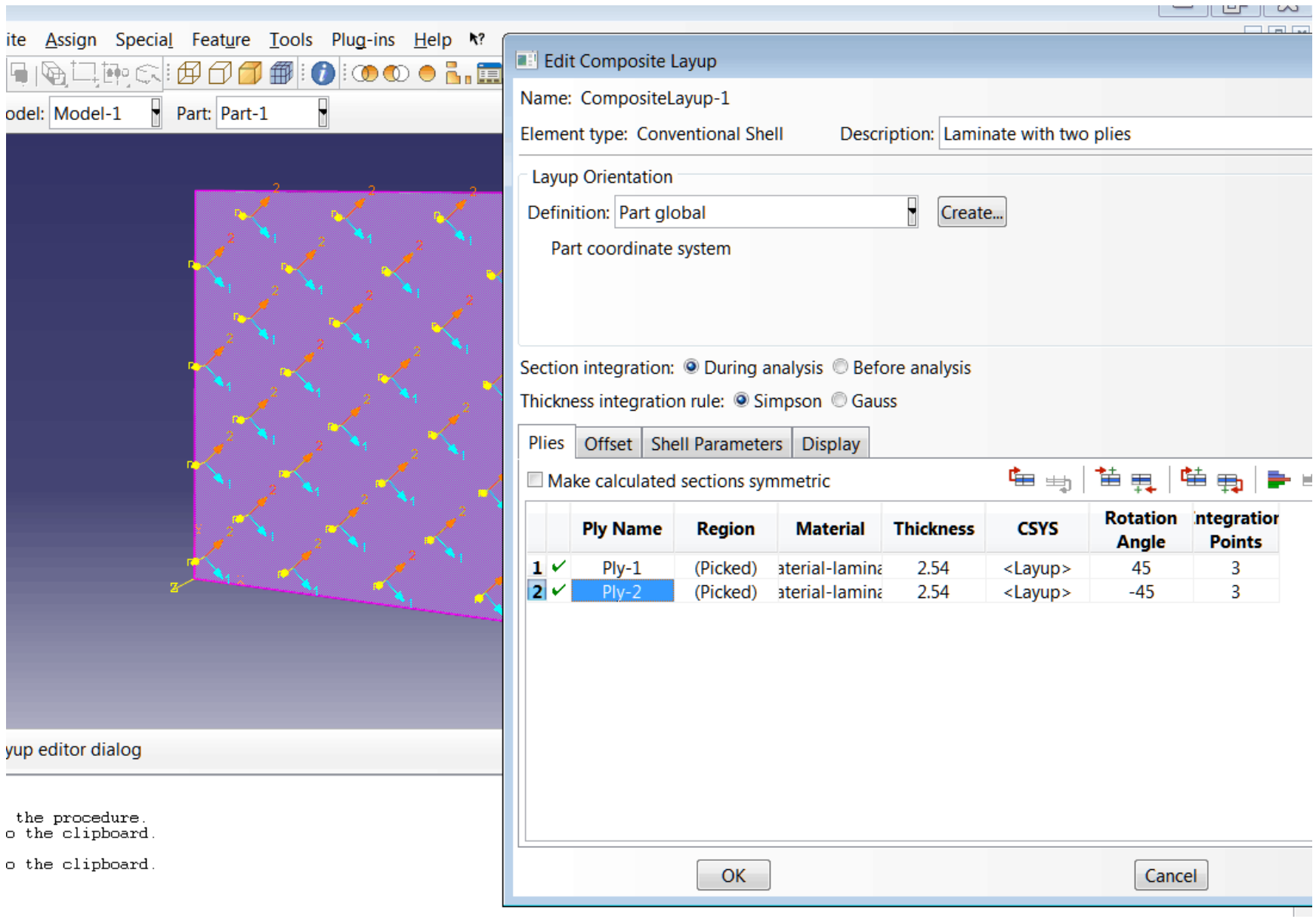


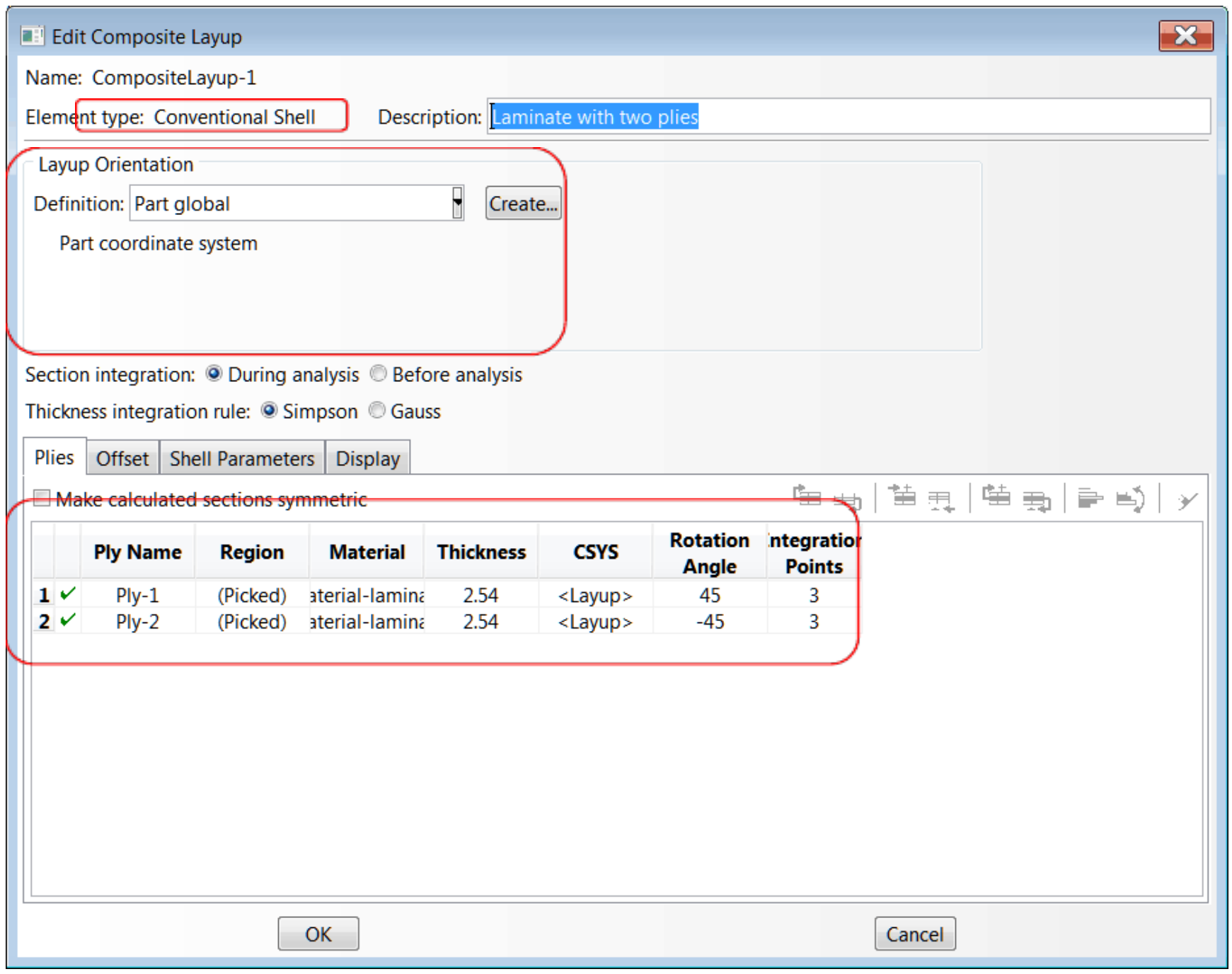


注意这三种单元类型的区别，详情请查看

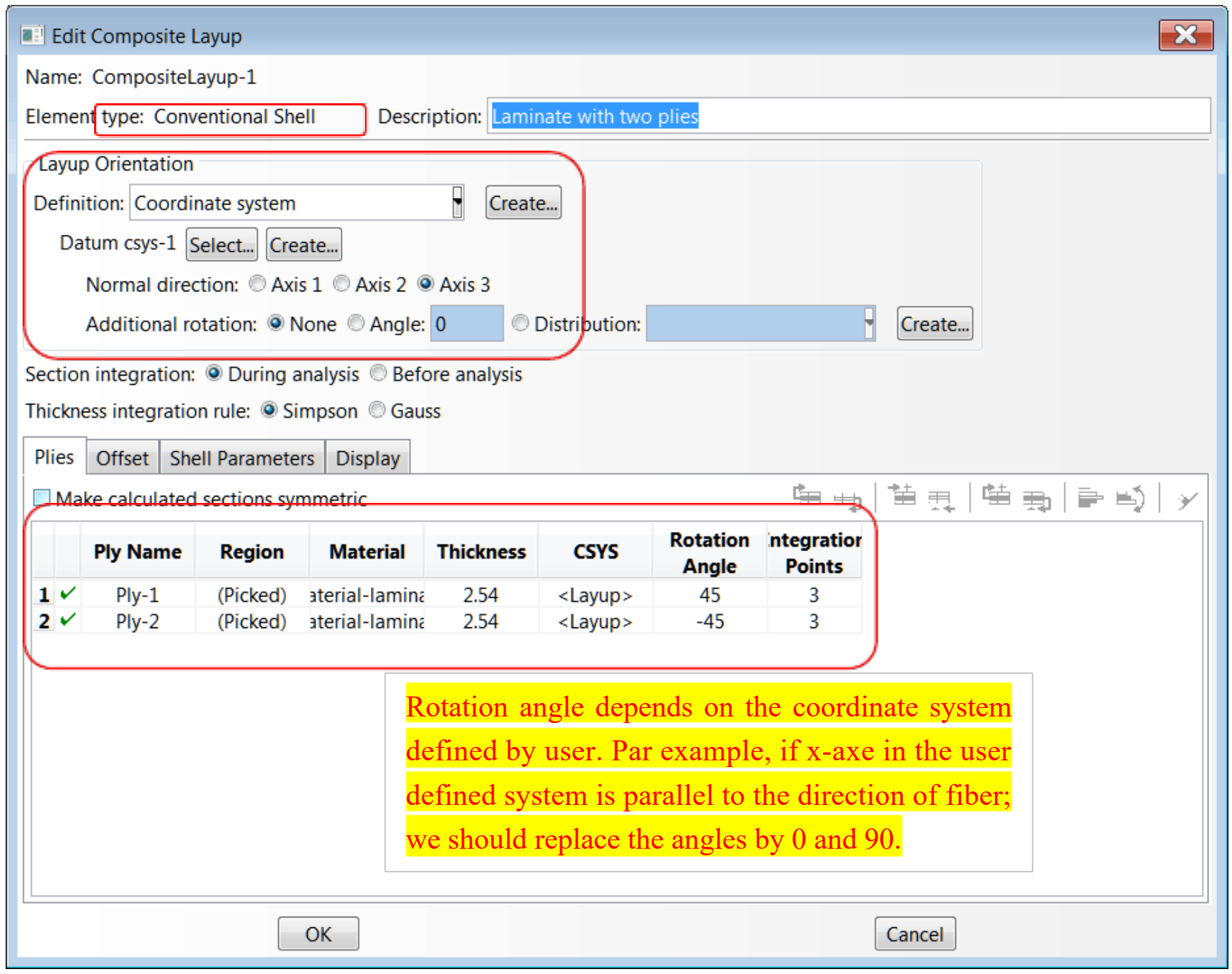
此处使用全局坐标系




ite Assign Special Feature Tools Plug-ins Help
odel: Model-1 Part: Part-1
yup editor dialog
the procedure.
o the clipboard.
o the clipboard.
Edit Composite Layup
Name: CompositeLayup-1
Element type: Conventional Shell
Description: Laminate with two plies
Layup Orientation
Definition: Part global
Create...
Part coordinate system
Section integration: During analysis Before analysis
Thickness integration rule: Simpson Gauss
Plies
Offset
Shell Parameters
Display
Make calculated sections symmetric
Ply Name
Region
Material
Thickness
CSYS
Rotation Angle
Integration Points
1 Ply-1 (Picked) aterial-lamina 2.54 <Layup> 45 3
2 Ply-2 (Picked) aterial-lamina 2.54 <Layup> -45 3
OK
Cancel


Edit Composite Layup
Name: CompositeLayup-1
Element type: Conventional Shell
Description: Laminate with two plies
Layup Orientation
Definition: Part global
Create...
Part coordinate system
Section integration: During analysis Before analysis
Thickness integration rule: Simpson Gauss
Plies
Offset
Shell Parameters
Display
Make calculated sections symmetric
Ply Name
Region
Material
Thickness
CSYS
Rotation Angle
Integration Points
1 Ply-1 (Picked) aterial-lamina 2.54 <Layup> 45 3
2 Ply-2 (Picked) aterial-lamina 2.54 <Layup> -45 3
OK
Cancel

使用用户自定义坐标系

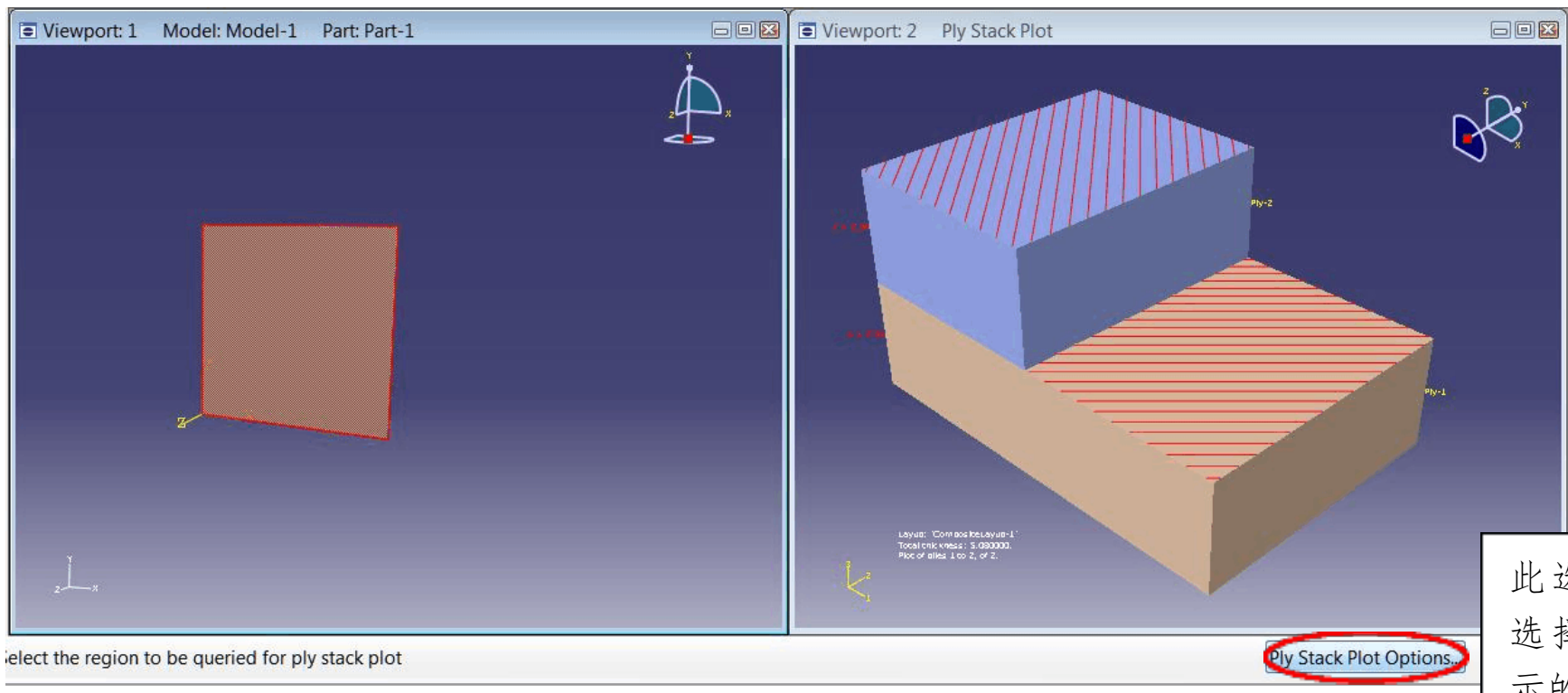


使用全局坐标系和局部坐标系的区别在下面这一步可以查看

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

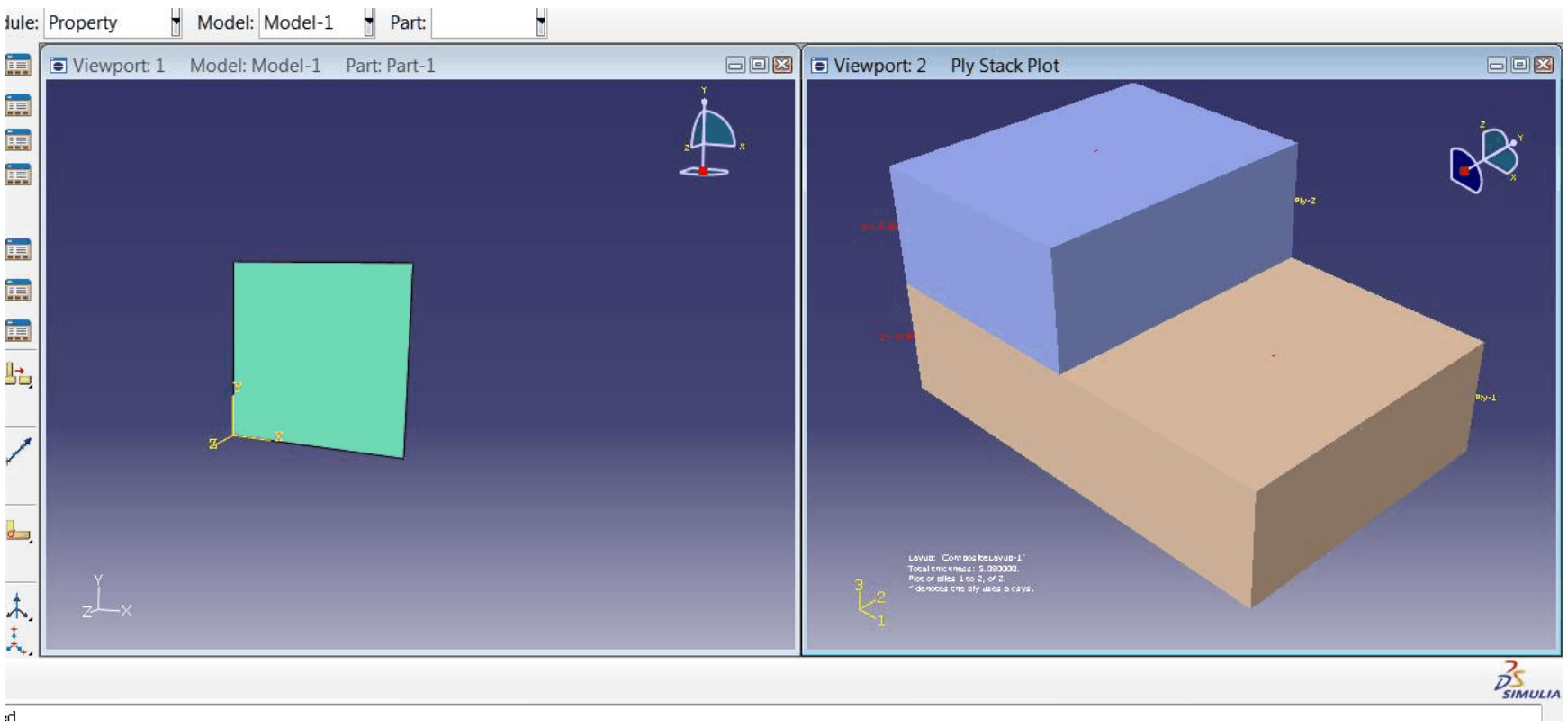
可以通过’工具——查询’来检查铺层（Tool ----Query----ply stack plot ）

Case 1 全局坐标系

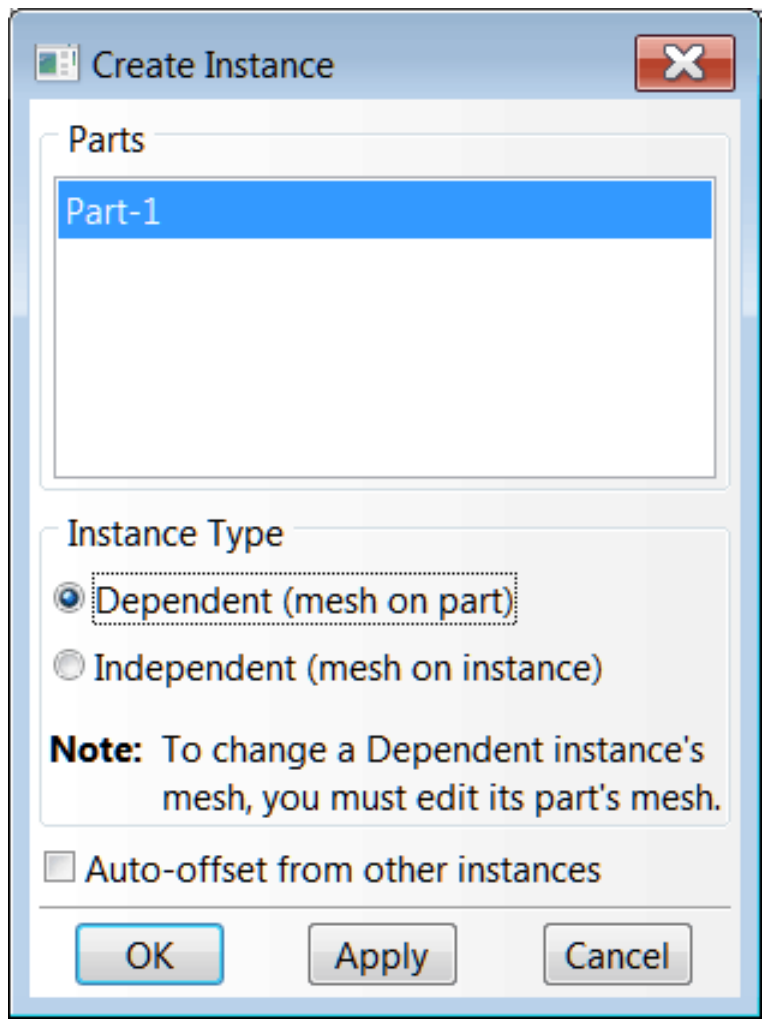


此选项可以选择图中显示的数据

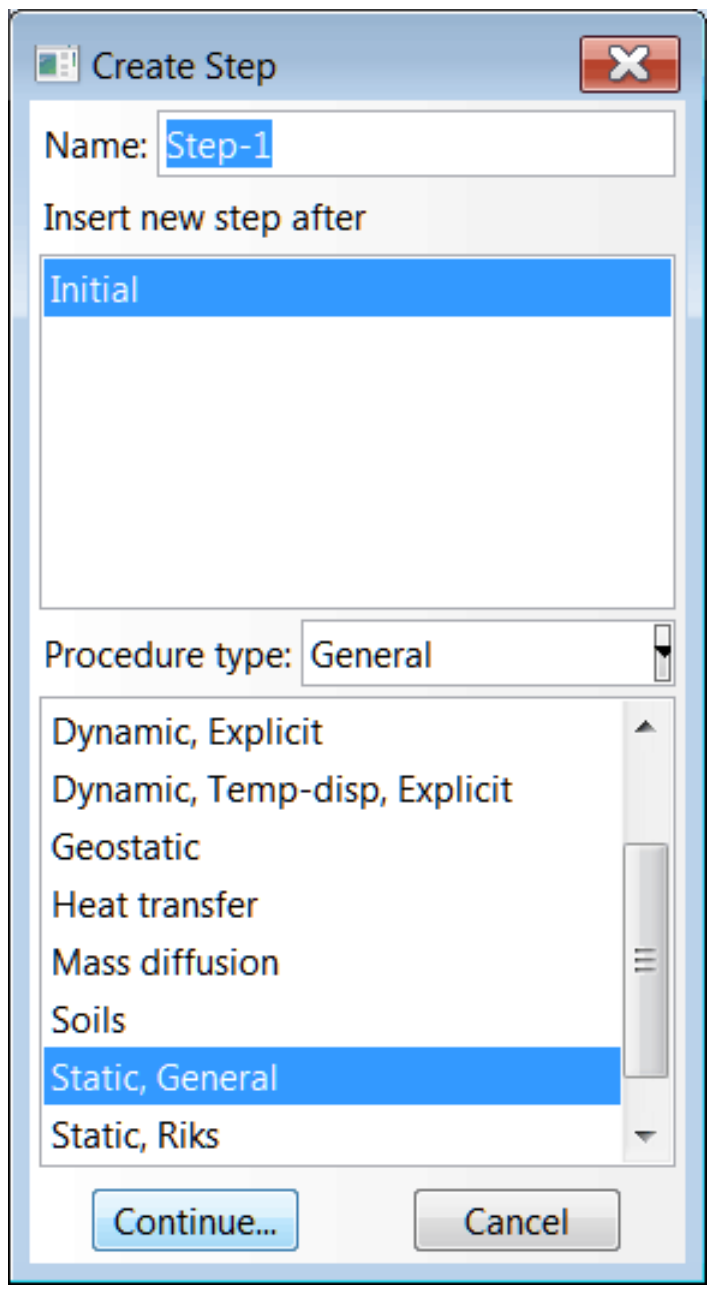
使用局部坐标系



生成装配件、定义分析步和输出要求

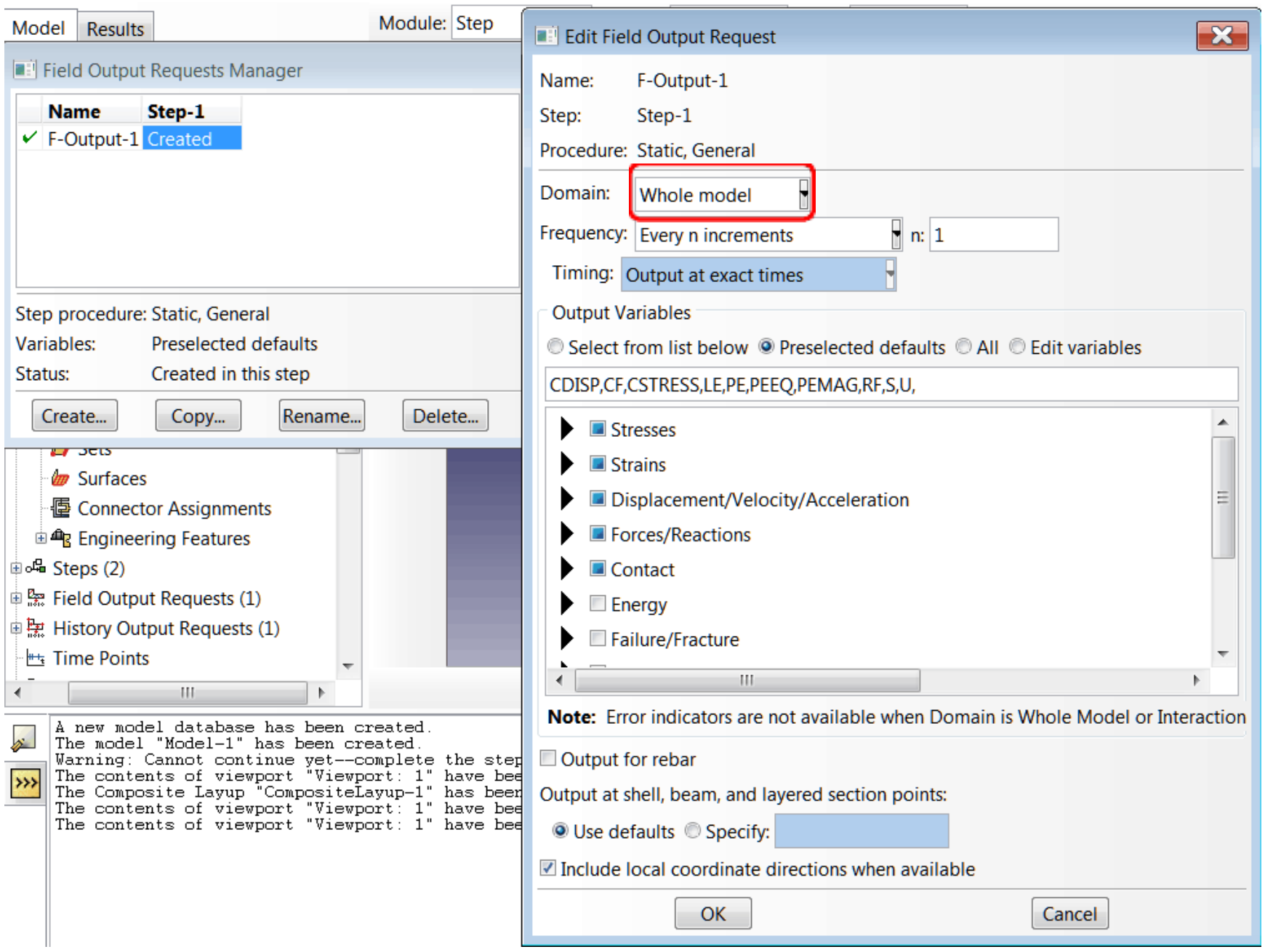


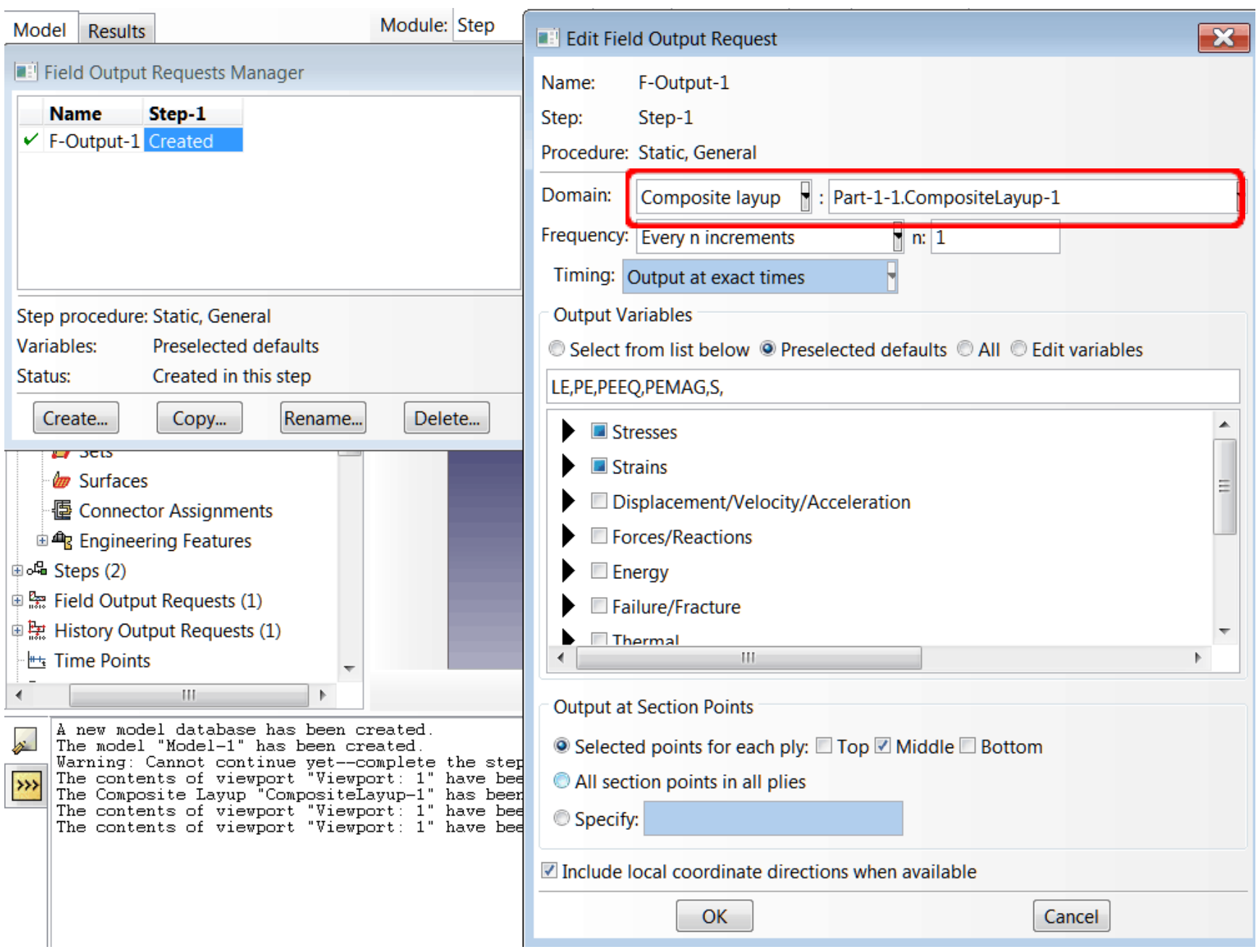
定义分析步，保留各项默认值即可。



场输出要求和历史输出要求都按默认的输出方式。

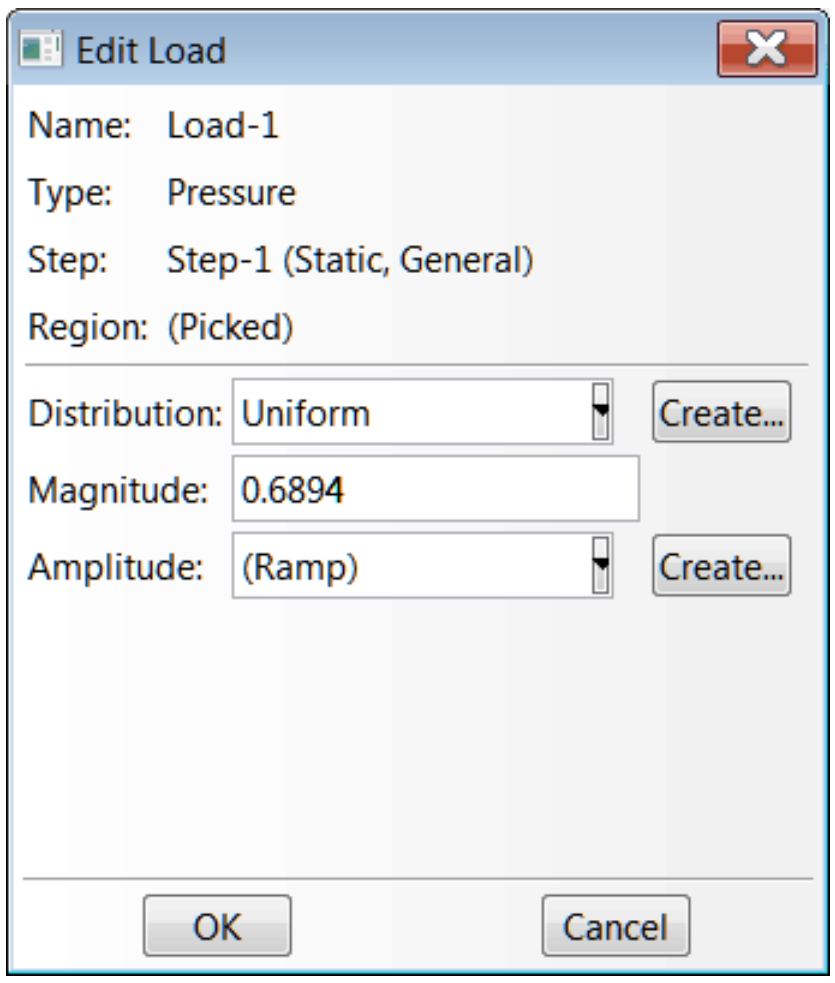
为了结果中能查看每个 ply 的输出情况，应该在场输出中进行一些修改



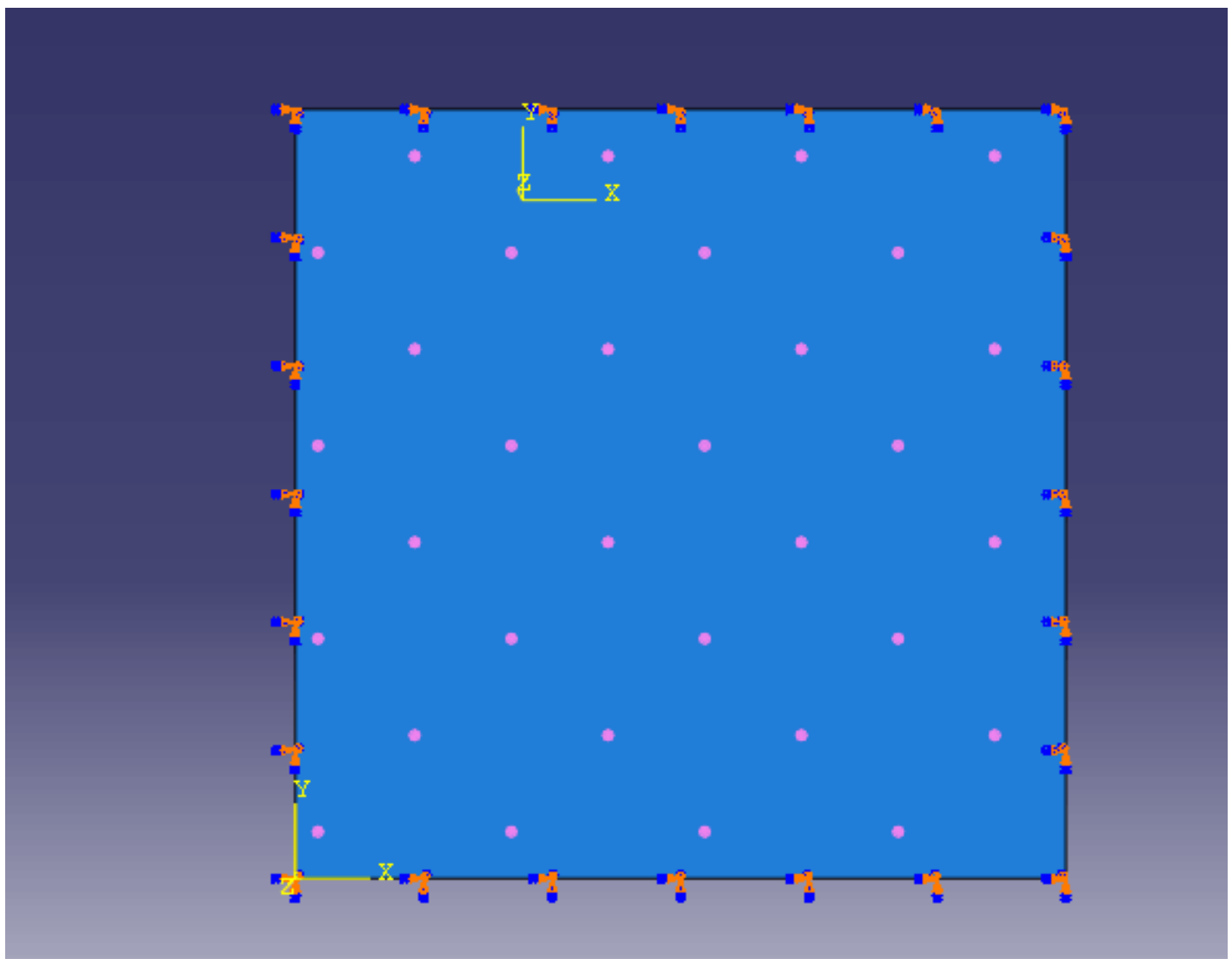


（场输出 field output 和历史输出 history output 的区别, refer to user s manual）

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



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：https://d.book118.com/128011116115007002