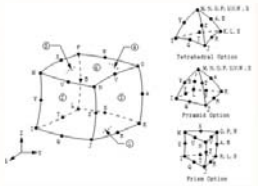
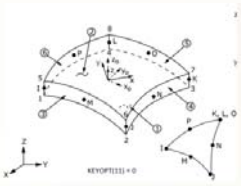

ANSYS WORKBENCH分析应用基础
LESSON21 杆梁单元和壳单元初步了解
课程制作 张 晔
QQ交流群：205237137

本课重点内容

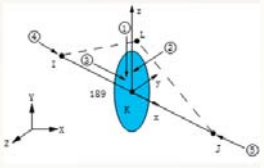
1. 初步了解三种单元
2. 杆梁单元及壳单元的分析基本操作
3. MESSAGE和帮助查看



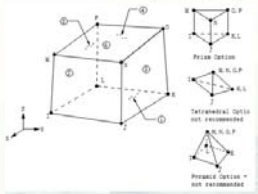
SOLID186（默认单元）



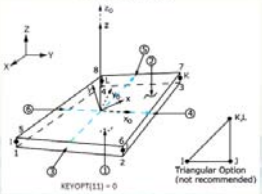
SHELL281



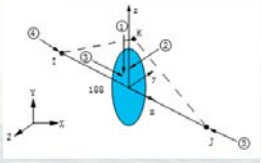
BEAM189



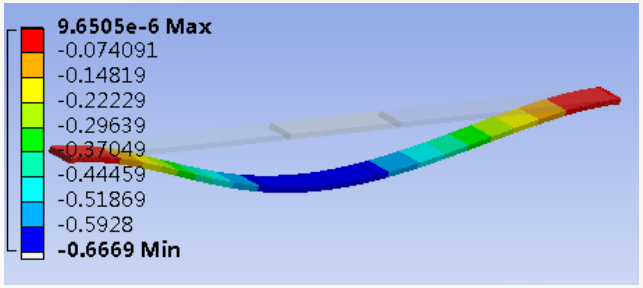
SOLID185

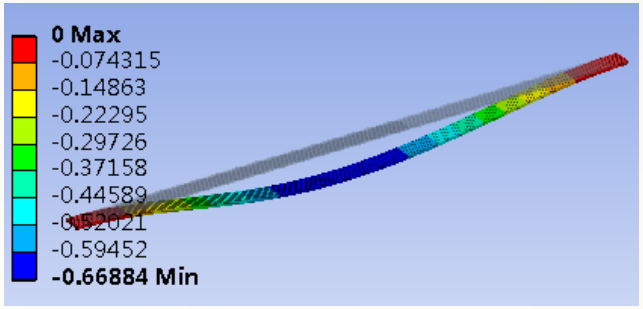


SHELL181（默认单元）



BEAM188（默认单元）

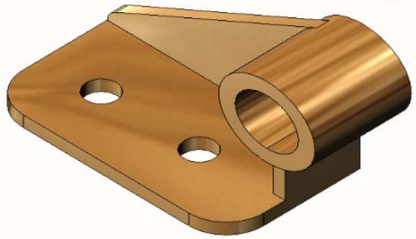


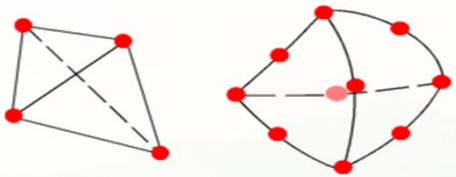



```
ELEMENT TYPE     1 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
 KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0

ELEMENT TYPE     2 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
 KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0

ELEMENT TYPE     3 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
 KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0
```

三种单元

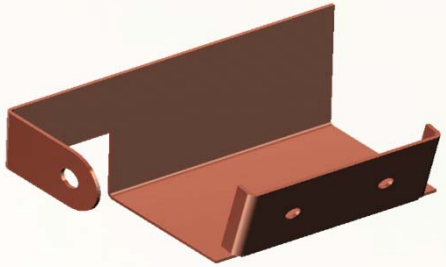
实体单元





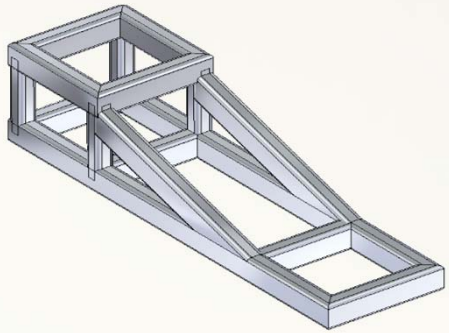
UX, UY, UZ

壳单元



UX, UY, UZ
ROTX, ROTY, ROTZ

杆/梁单元



UX, UY, UZ
ROTX, ROTY, ROTZ

单元分类

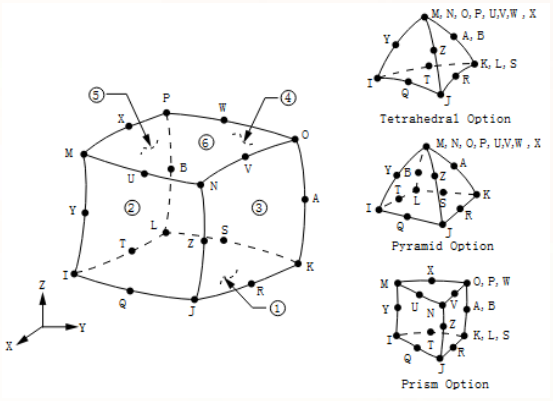
根据单元的形状分类，可以分为实体单元、壳单元和杆梁单元；

其次实体单元可分为六面体单元和四面体单元，壳单元可以分为四边形单元和三角形单元；

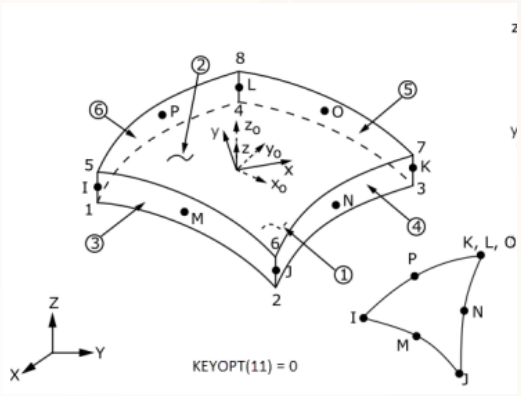
再根据单元的形函数复杂程度又可以分为一阶单元和二阶单元。

最终单元分类大致有二阶六面体单元（退化为二阶四面体单元）、一阶六面体单元（退化为一阶四面体单元）、二阶四边形单元（退化为二阶三角形单元）、一阶四边形单元（退化为一阶三角形单元）、二阶杆梁单元和一阶杆梁单元。

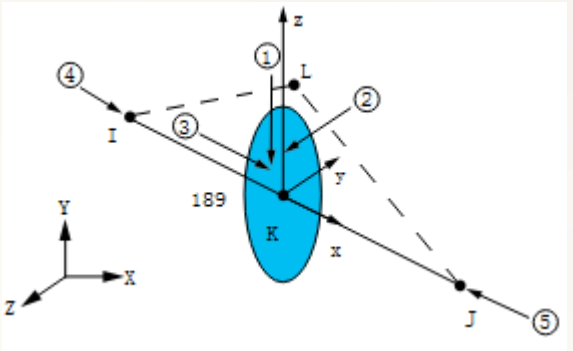
WORKBENCH中所使用的单元



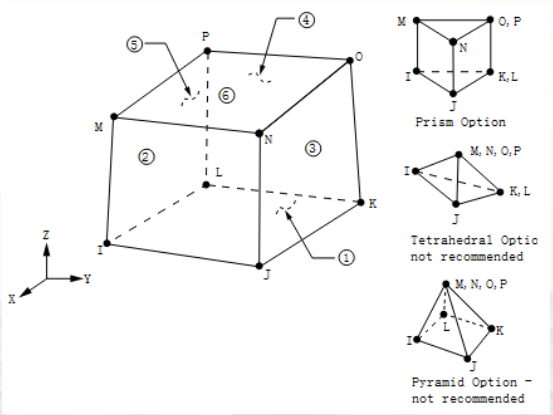
SOLID186（默认单元）



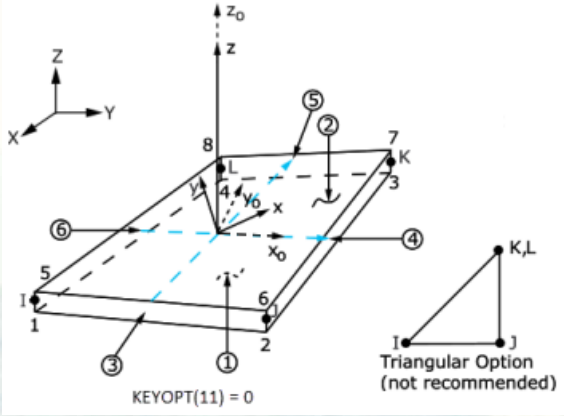
SHELL281



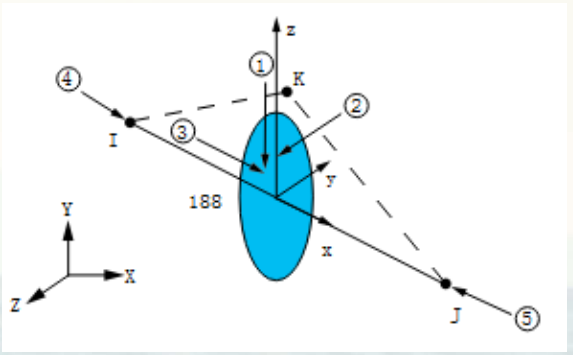
BEAM189



SOLID185



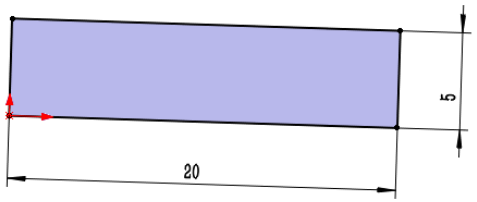
SHELL181 （默认单元）

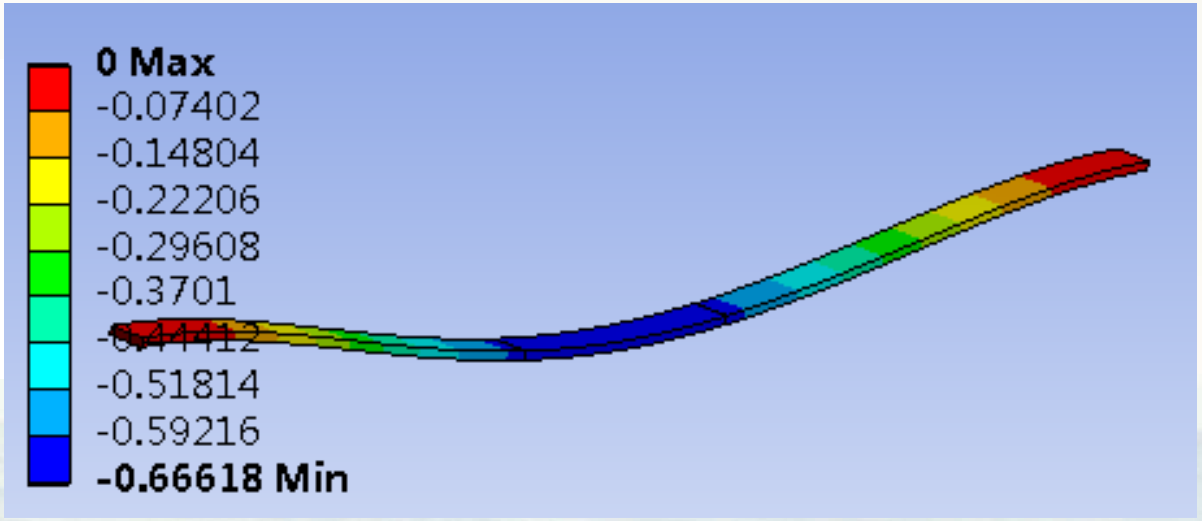


BEAM188 （默认单元）

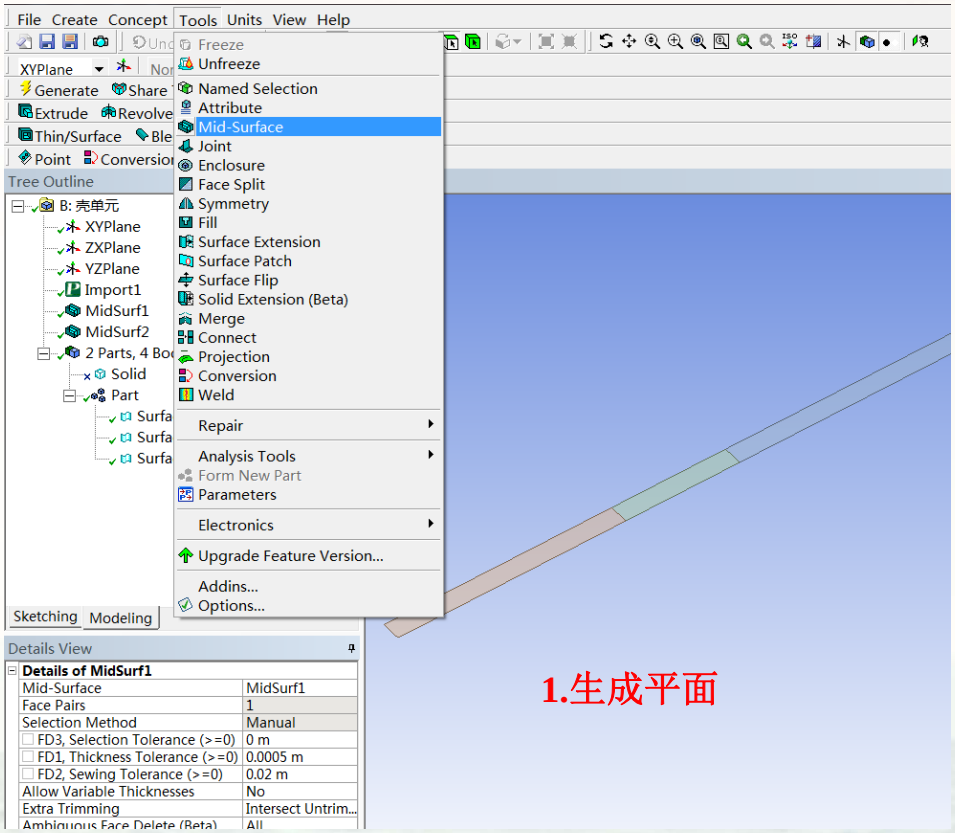
实体单元的计算结果

横梁全长1m，物体长度0.2m，放置在横梁中间，横梁两端全约束，横梁材料和物体材料的弹性模量为2.1E11Pa，横梁材料密度为7800kg/m3，物体材料密度为5100kg/m3（物体自重2N），在考虑梁自重的情况下，考察横梁中点挠度。（目前为配合壳单元和梁单元的对比，以及后期自由度的讲解，本课中我们将不使用对称模型而使用全模型进行计算）

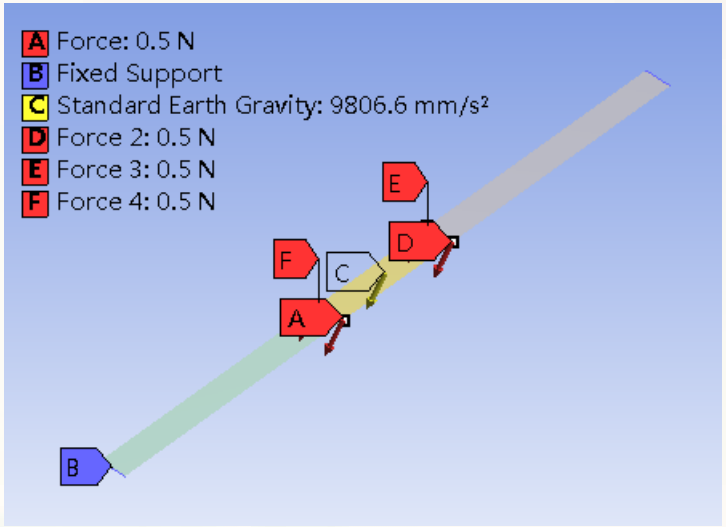




壳单元的操作过程

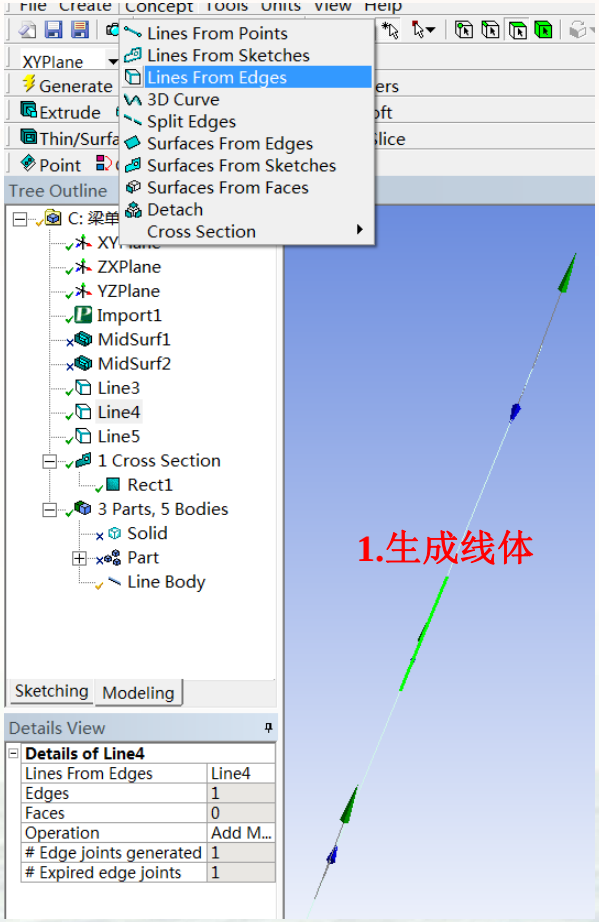


1.生成平面

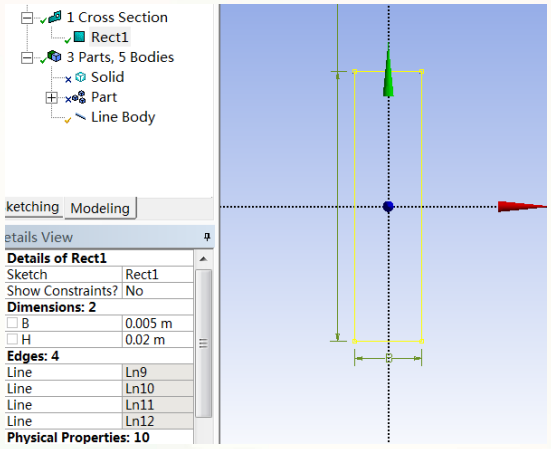


2.施加边界条件

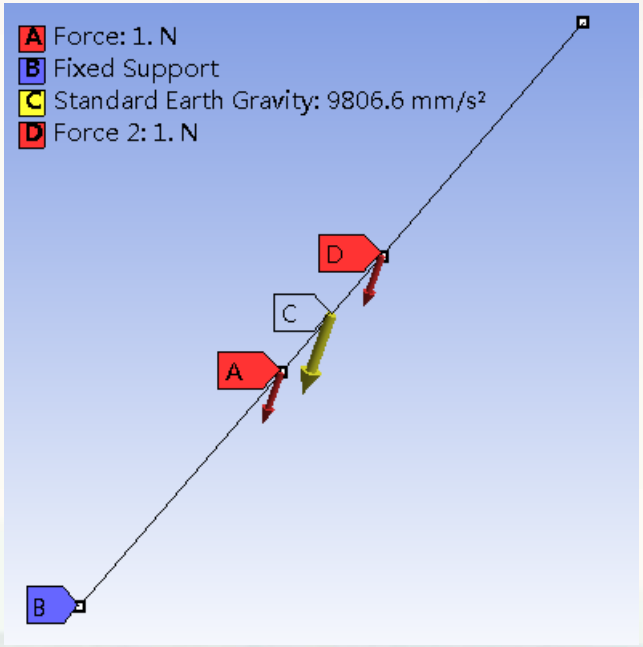
梁单元的操作过程



1.生成线体

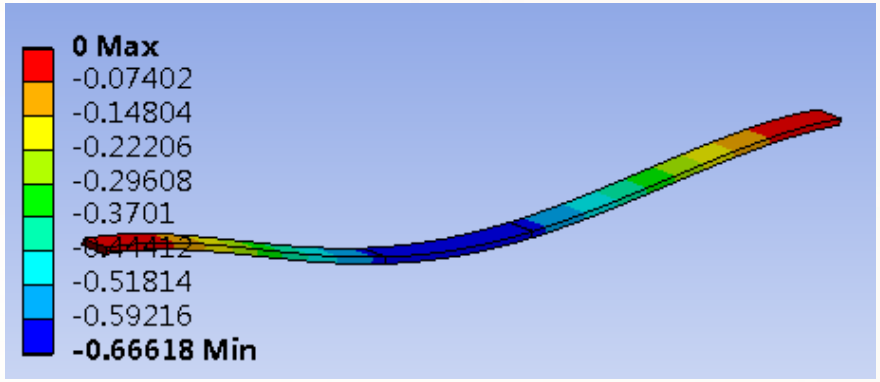


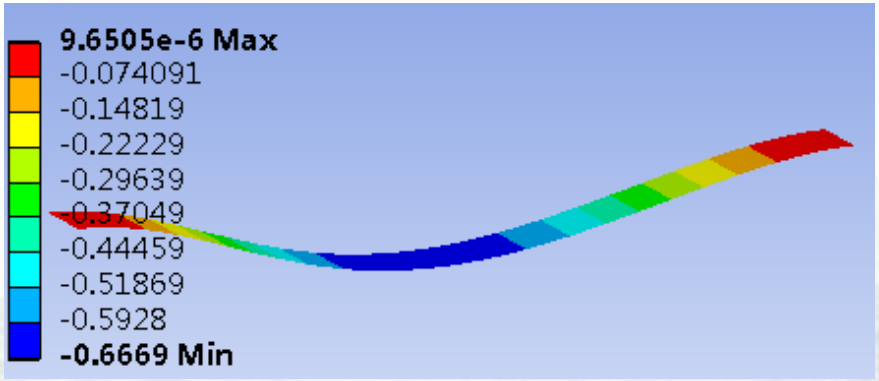
2.生成截面

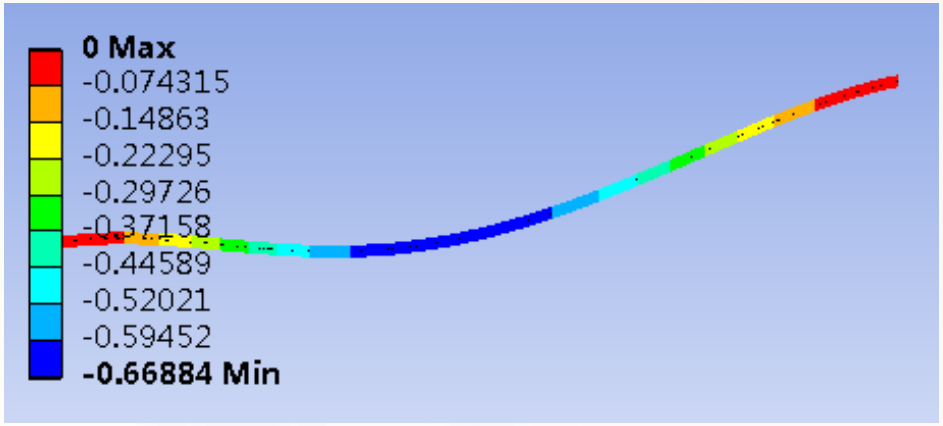


3.施加边界条件

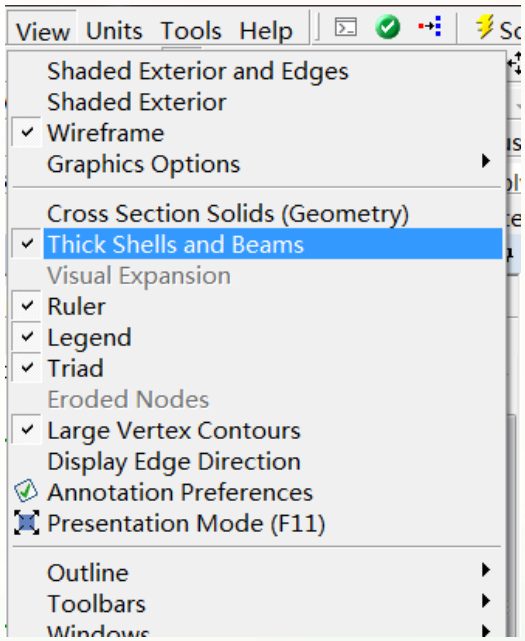
计算结果对比

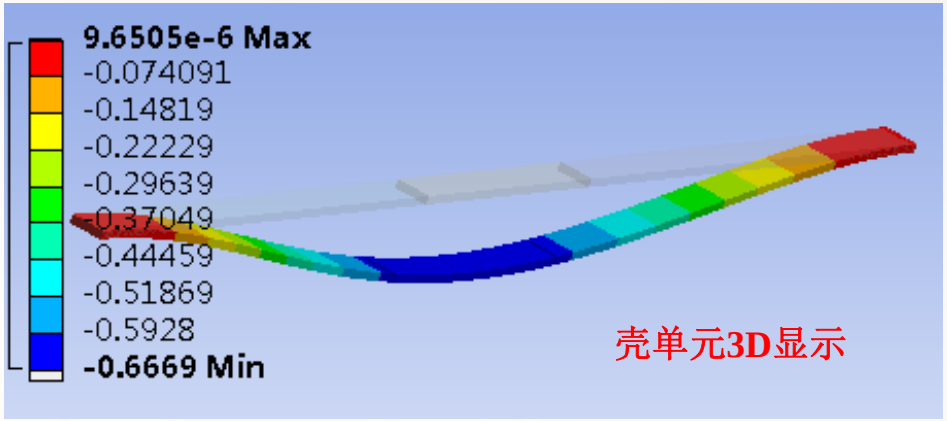

0 Max
-0.07402
-0.14804
-0.22206
-0.29608
-0.3701
-0.44412
-0.51814
-0.59216
-0.66618 Min

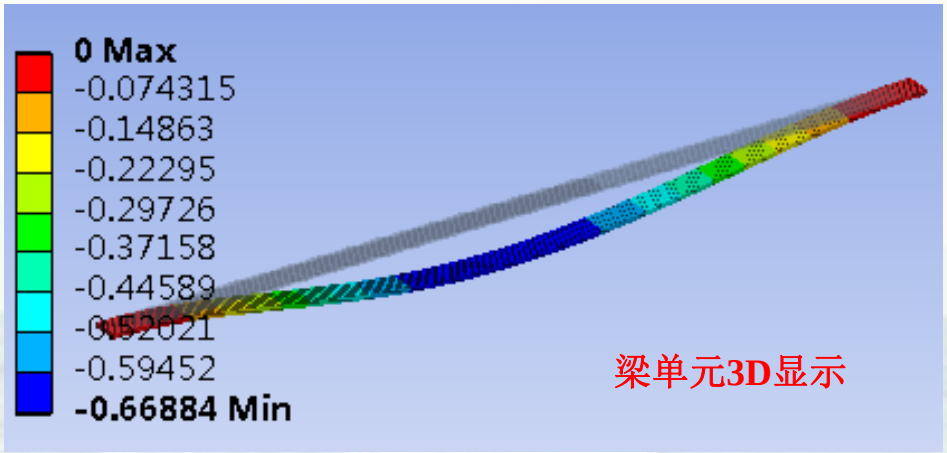

9.6505e-6 Max
-0.074091
-0.14819
-0.22229
-0.29639
-0.37049
-0.44459
-0.51869
-0.5928
-0.6669 Min


0 Max
-0.074315
-0.14863
-0.22295
-0.29726
-0.37158
-0.44589
-0.52021
-0.59452
-0.66884 Min

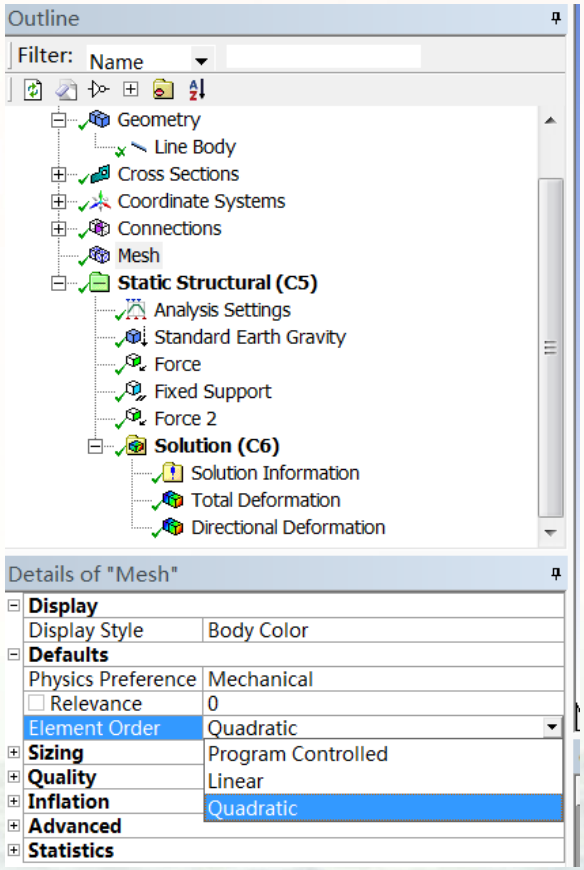
梁单元和壳单元的结果3D显示

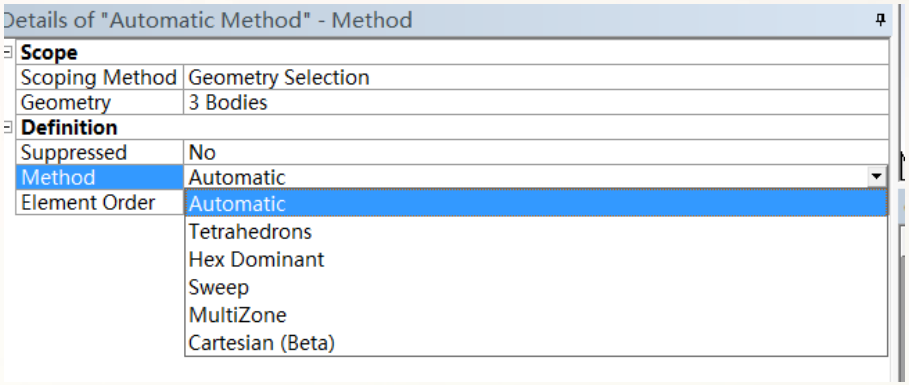






ANSYS选择单元查看






```
ELEMENT TYPE    1 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
 KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0

ELEMENT TYPE    2 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
 KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0

ELEMENT TYPE    3 IS SOLID185. IT IS ASSOCIATED WITH LINEAR MATERIALS ONLY
AND POISSON'S RATIO IS NOT GREATER THAN 0.49. KEYOPT(2)=3 IS SUGGESTED AND
HAS BEEN RESET.
 KEYOPT(1-12)=    0    3    0    0    0    0    0    0    0    0    0    0
```

本课的一个重要说明

原则上，任何模型都可以用实体网格来进行划分，但是有些时候由于模型形状导致网格计算量庞大，我们会将薄板模型简化成壳单元，细长杆件模型简化成壳单元或者杆梁单元进行求解计算。


下一期视频，我将和大家一起交流关于
《再论自由度问题》