



清华大学

Tsinghua University

基于APDL和Python语言的 工型截面简支梁整体稳定性能分析

土木与建筑工程CAE课程汇报

刘晓玲 2015310063

钢结构与玻璃幕墙课题组

2016. 06. 02



◎ 目录

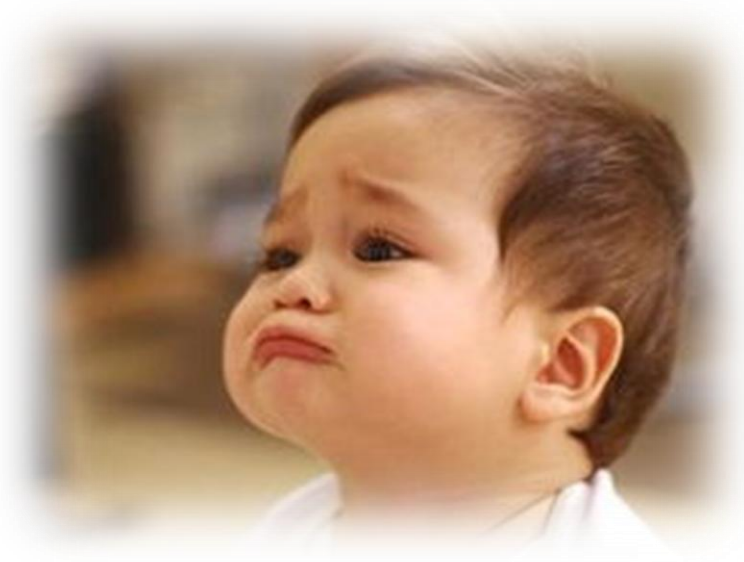
- 课题背景
- 课题内容
 - ANSYS、ANSYS Workbench、Abaqus
 - 二次开发简介
 - APDL参数化建模过程
 - Python参数化建模过程
- 课题总结





◎ 1. 课题背景

ANSYS



ABAQUS

◎ 2.1 软件对比

● ANSYS

➤ ANSYS经典版（Mechanical APDL）（17.0）

➤ **ANSYS Workbench（15.0?）**

- ✓ 多场耦合分析（项目流程图）
- ✓ 流场分析（CFX, FLUENT），电磁场分析
- ✓ 工作界面改善
- ✓ 简化建模分析过程



● ABAQUS（2016）

- ✓ 界面友好，强调有限元分析过程
- ✓ 非线性求解功能
- ✓ 建模组装复杂





◎ 2.1 软件对比

● 软件对比

- 市场占有率： ANSYS>ABAQUS
- 结构方面：
 - ✓ 接触问题：方便程度和收敛顺序 ABAQUS>ANSYS
 - ✓ 结构优化设计或拓扑优化设计： ANSYS>ABAQUS
 - ✓ 多场耦合分析： ANSYS>ABAQUS
 - ✓ 界面菜单建模： ABAQUS>ANSYS
 - ✓ 编程参数建模： ANSYS>ABAQUS
 - ✓ 结构网格划分的方便程度： ABAQUS>ANSYS

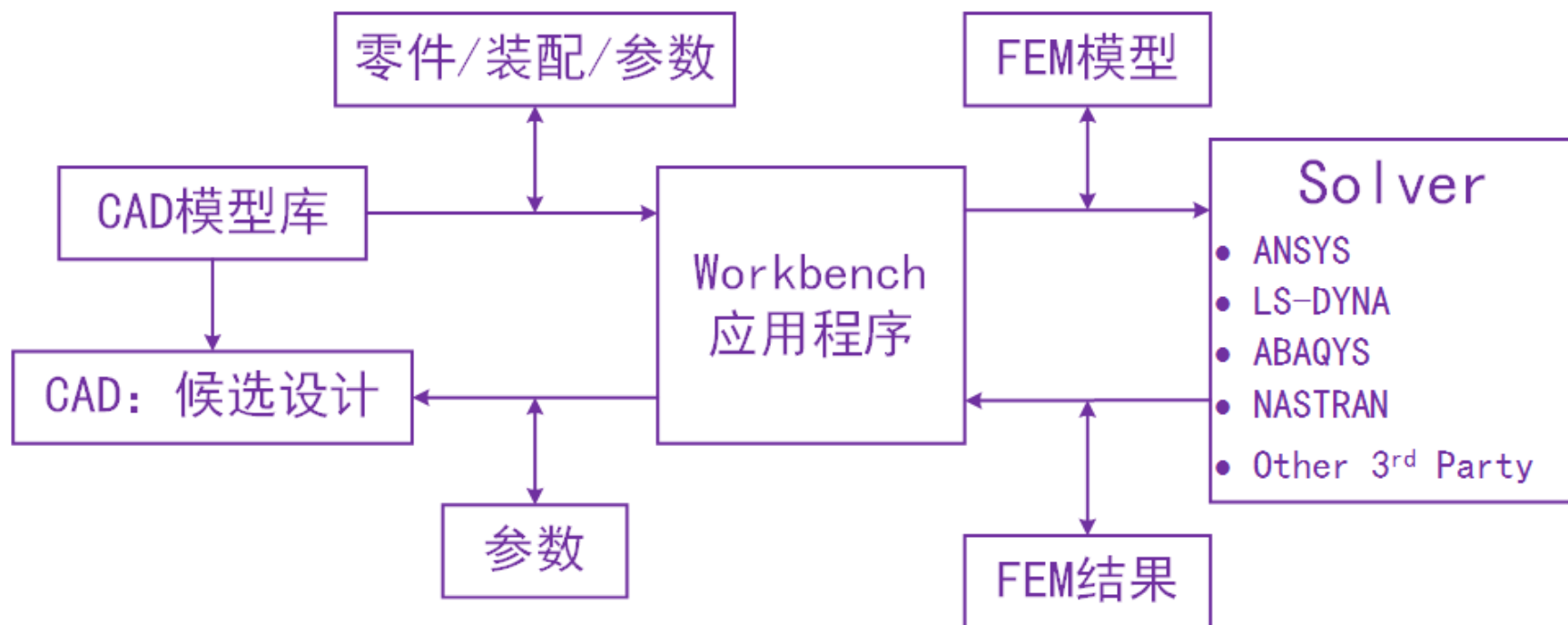


◎ 2.1 ANSYS Workbench

- **ANSYS Workbench（协同仿真技术）**
 - CAD软件和模型数据的整合
 - CAE软件和模型数据的整合
 - CAD/CAE数据共享与交流
- **新功能**
 - ✓ AWE（ANSYS Workbench Environment）
 - ✓ DesignModeler建模工具
 - ✓ CAD双向参数链接互动
 - ✓ 项目数据自动跟新
 - ✓ 全面的参数管理
 - ✓ 无缝集成的优化设计工具

2.1 ANSYS Workbench

● ANSYS Workbench（协同仿真技术）



基于Workbench协同环境的分析数据流



◎ 2.1 ANSYS Workbench

● ANSYS Workbench

➤ 优势

- ✓ 与CAD软件接口
- ✓ 模型处理能力：复杂接触问题
- ✓ 网格划分能力（六面体单元）
- ✓ 后处理功能
- ✓ 经典界面——高级研究人员
- ✓ Workbench——一般工程师

◎ 2.2 二次开发

● ANSYS二次开发技术



➤ 良好开放性

➤ 主要开发工具

参数化程序设计语言（APDL）
用户界面设计语言（UIDL）
用户可编程程序特性（UPFs）
专用界面开发工具（TCL/TK）

● ANSYS Workbench二次开发技术

➤ 各个模块都有其特有的脚本语言

➤ 结构模块下就需要XML语言，Python语言和APDL语言



◎ 2.2 二次开发

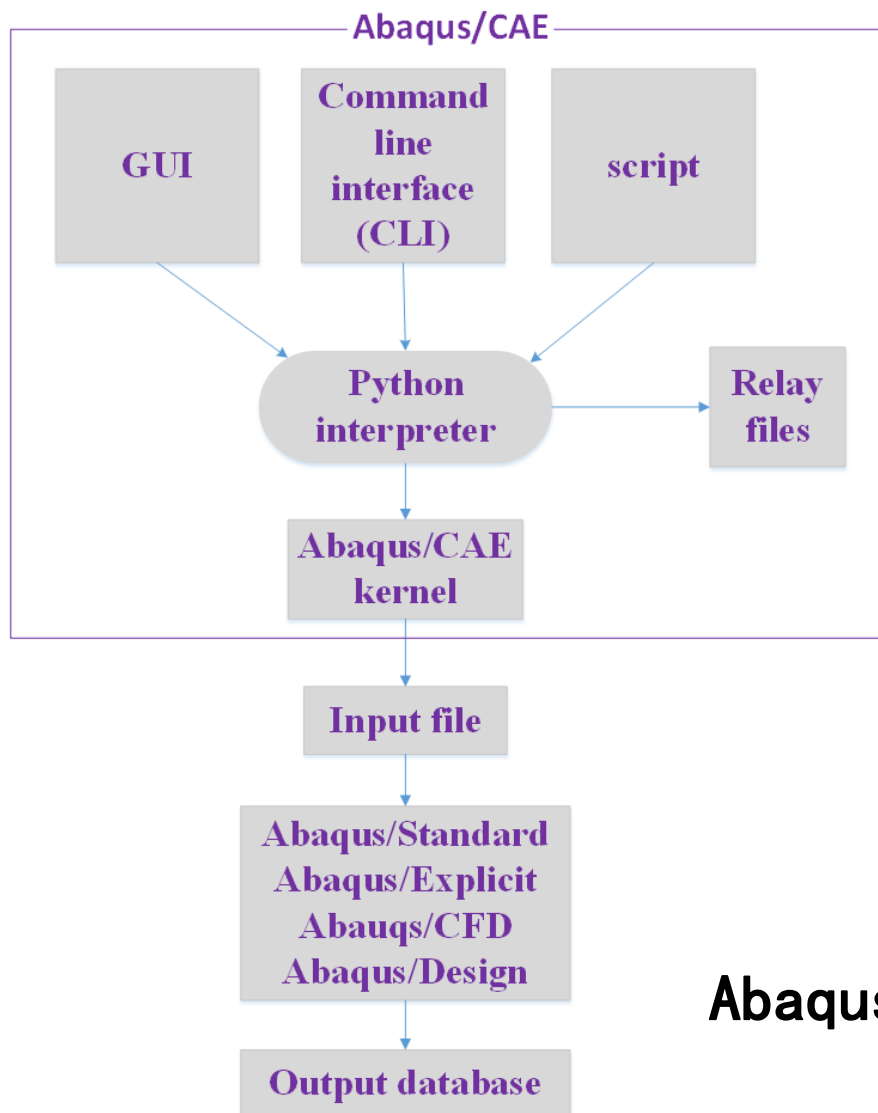
● ABAQUS二次开发技术



- 接口 { 子程序接口 (user subroutine)
脚本接口 (Scripting interface)

- 主要开发方法 { Python脚本语言
Fortran子程序
Plug-in 插件开发功能
其他软件平台, 如PyQt

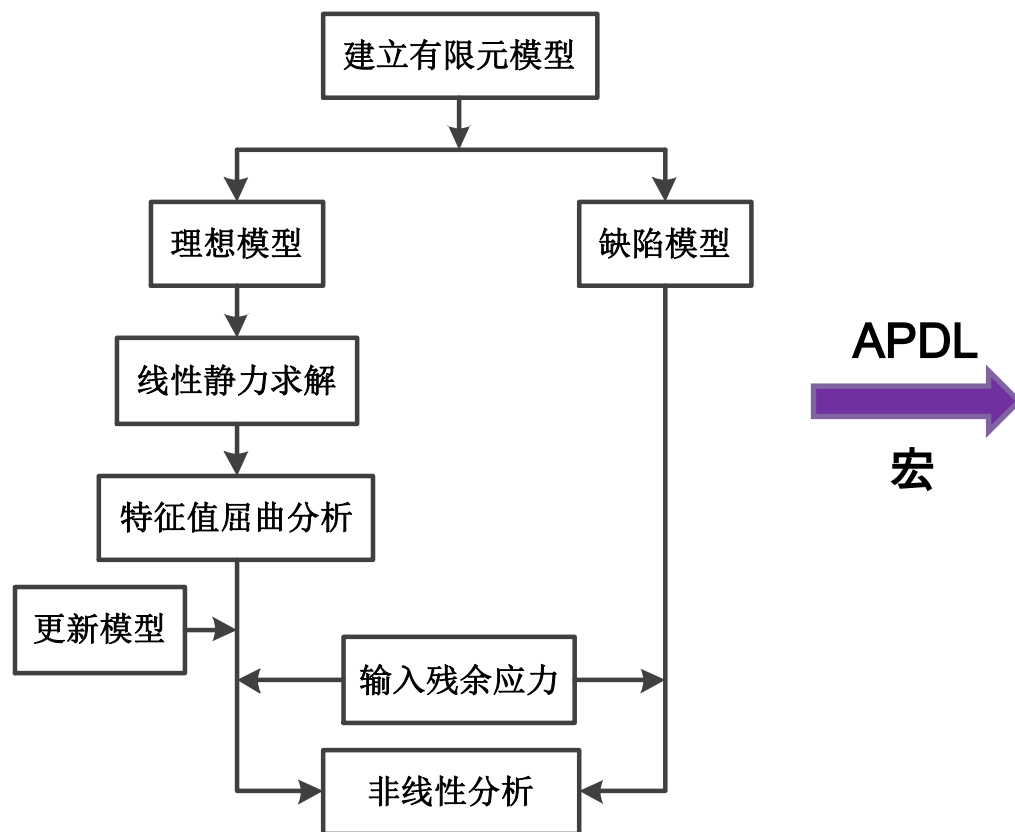
2.2 二次开发



Abaqus中Python脚本的通信关系图

2.3 APDL参数化建模

● APDL (ANSYS Parametric Design Language)



参数分析:

参数化建模

参数化的网格划分与控制

参数化的材料定义

参数化载荷

参数化边界条件定义

参数化的分析控制

参数化求解

参数化后处理结果的显示

2.3 APDL参数化建模

```

/*****
/FILENAME,BD-1,0
/CONFIG,NRES,3000
/PREP7
/*****
!====Component geometry
/*****
l=5600      !span
a=1800      !bending and shear district
ext=200     !extent distance
ltotal=l+2*ext !total distance

b=200       !flange width
h=500       !section height

tf0=13      !min flange thickness
m=0.00583   !change rate of flange thickness
tf=tf0+m*a  !max flange thickness
tw=12       !web thickness

ts=12       !stiff plate thickness
th=8        !welding size

bf=(b-tw)/2 !extent flange width
hw=h-2*tf   !web height

/*****
!====element size
/*****
fl_div=4    ! number of division for flanges (should be even)
web_div=10  ! number of division for web (default value is 15)

mmm=50      ! division of flange
lll=50      ! division of height of web
nnn=50      ! division along beam
uou=1*nnn   ! division alone beam coarse parts

```

```

/*****
!====ending element and material attribute
/*****

E=206000    !component elastic modulus
NOU=0.3      !component poisson ratio

ET,1,SHELL181 !beam
ET,2,SOLID185 !dianban

MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,EX,1,,E
MPDATA,PRXY,1,,NOU

TB,MISO,1,1,4,0 !Q345
TBTEMP,0
TBPT,,0,0
TBPT,,345/E,345
TBPT,,0.02,345
TBPT,,0.2,480

```

变量定义及材料参数定义

PSPad.exe 文本编辑器
. mac

2.3 APDL参数化建模

```

/=====
/===changing sections
/=====

/=====sections
*DIM,TFL,TABLE,6,2,1,Z,X
*SET,TFL(0,1,1),0
*SET,TFL(0,2,1),b
*SET,TFL(1,0,1),-ext
*SET,TFL(1,1,1),tf0
*SET,TFL(1,2,1),tf0
*SET,TFL(2,0,1),0
*SET,TFL(2,1,1),tf0
*SET,TFL(2,2,1),tf0
*SET,TFL(3,0,1),a
*SET,TFL(3,1,1),tf
*SET,TFL(3,2,1),tf
*SET,TFL(4,0,1),1-a
*SET,TFL(4,1,1),tf
*SET,TFL(4,2,1),tf
*SET,TFL(5,0,1),1
*SET,TFL(5,1,1),tf0
*SET,TFL(5,2,1),tf0
*SET,TFL(6,0,1),1+ext
*SET,TFL(6,1,1),tf0
*SET,TFL(6,2,1),tf0

sect,1,shell,,                                !top flange
secddata,0,1,0,5
secoffset,TOP
secf,%TFL%,0
seccontrol,0,0,0,0,1,1,1

sect,2,shell,,                                !bottom flange
secddata,0,1,0,5
secoffset,BOT
secf,%TFL%,0
seccontrol,0,0,0,0,1,1,1

sect,3,shell,,                                !web
secddata,tw,1,0,5
secoffset,MID
seccontrol,0,0,0,0,1,1,1

sect,4,shell,,                                !stiff plate
secddata,ts,1,0,5
secoffset,MID
seccontrol,0,0,0,0,1,1,1

```

变厚度壳定义



2.3 APDL参数化建模

```
!====Geometry modeling
```

```
! extentions
```

```
k,1,0,0,-ext
```

```
k,2,0,0,0
```

```
k,3,b/2,0,-ext
```

```
k,4,b/2,0,0
```

```
k,5,b,0,-ext
```

```
k,6,b,0,0
```

```
k,7,0,h,-ext
```

```
k,8,0,h,0
```

```
k,9,b/2,h,-ext
```

```
k,10,b/2,h,0
```

```
k,11,b,h,-ext
```

```
k,12,b,h,0
```

```
! extentions
```

```
a,2,4,3,1
```

```
a,4,6,5,3
```

```
a,8,10,9,7
```

```
a,10,12,11,9
```

```
a,4,3,9,10
```

几何模型

```
!====define meshing size
```

```
! Top flange
```

```
!=====
```

```
Lsel,s,loc,x,0,b
```

```
Lsel,r,loc,y,h
```

```
Lsel,r,loc,z,-ext
```

```
Lesize,all,mmm
```

```
!====meshing
```

```
asel,all
```

```
! ===== top flanges
```

```
asel,all
```

```
TYPE,1
```

```
MAT,1
```

```
REAL,
```

```
ESYS,0
```

```
SECNUM,1
```

```
asel,s,loc,x,0,b
```

```
asel,r,loc,y,h
```

```
asel,r,loc,z,-ext,1+ext
```

```
amesh,all
```

网格划分



2.3 APDL参数化建模

```
!=====
!===define component
!=====
nse1,s,loc,x,0,b
nse1,r,loc,y,h
nse1,r,loc,z,a
cm,Loads13,node
nse1,all

nse1,s,loc,x,0,b
nse1,r,loc,y,h
nse1,r,loc,z,l-a
cm,Loads23,node
nse1,all

!=====
!===Applying Load
!=====

cmse1,s,loads13
d,all,uy,-0.1
nse1,all

cmse1,s,loads23
d,all,uy,-0.1
nse1,all
```

施加位移荷载

```
!=====
!===Applying constraints
!=====
!supporting constraints
nse1,s,loc,x,0,b
nse1,r,loc,y,0
nse1,r,loc,z,0
d,all,ux,,,,uy
nse1,all

!=====
!=== buckling solving
!=====

/solu
antype,0,

solve
finish

/solu
antype,1,

bucopt,LANB,5,,,range

solve
save
finish
```

特征值屈曲分析



2.3 APDL参数化建模

```
!===== updating model =====  
/PREP7  
nset,all  
upgeom,ogi,1,mdnum,BD-1,rst      ! load step, substep, file name-----  
nset,all  
eset,all  
  
! =====  
!=====Residual Stress  
! =====  
  
! sfrte: Residual Stress flange in tension at edge  
  
*if,ffy,le,800,then  
sfrte=50  
*else  
sfrte=.3*ffy  
*endif  
  
! sfrt: Residual Stress flange in tension at center  
  
*if,ffy,le,345,then  
sfrt=ffy  
*elseif,ffy,gt,345,and,ffy,lt,550,then  
sfrt=345  
*elseif,ffy,ge,550,then  
sfrt=460  
*endif
```

更新几何模型

定义并施加残余应力



2.3 APDL参数化建模

```
!=====Nonlinear analysis=====
/PREP7
esel,all
nsel,all

cmsel,s,loads13          ! (loads13) (loads12) (loads23)
d,all,uy,-65
nsel,all

cmsel,s,loads23          ! (loads13) (loads12) (loads23)
d,all,uy,-65
nsel,all

nsel,all
esel,all

! nonlinear information
/solu
antype,0,
nlgeom,on
!arclen,on

!neqit,50
outres,all,1
deltim,0.005      ! time step size

solve
```

位移加载
大变形开关
弧长法
步长



2.3 APDL参数化建模

```
/POST26
NUMVAR,200

!get the reforce of support
NSEL,S,LOC,Z,a
NSEL,R,LOC,X,0,B
NSEL,R,LOC,Y,H
*GET,NCOUNT,NODE,,COUNT
NNOW=0

*DO,I,1,NCOUNT
NNOW=NDNEXT(NNOW)
RFORCE,3,NNOW,F,Y,P-LOAD
ADD,4,4,3,,,,,-1
*ENDDO

!get the vertical displacement of middle span
NSEL,S,LOC,Z,L/2
NSEL,R,LOC,X,0,B
NSEL,R,LOC,Y,0
*GET,NCOUNT,NODE,,COUNT
NNOW=0
NSOL,2,NDNEXT(NNOW),U,Y,DISPLACEMENT
ADD,2,2,,,,,-1

!get the lateral displacement of middle span
NSEL,S,LOC,Z,L/2
NSEL,R,LOC,X,0,B
NSEL,R,LOC,Y,0
*GET,NCOUNT,NODE,,COUNT
NNOW=0
NSOL,5,NDNEXT(NNOW),U,X,DISPLACEMENT
ADD,5,5,,,,,1
```

plot load-vertical displacement curve

```
VAR,2
AXLAB,X,v-DISPLACEMENT
AXLAB,Y,LOAD
LVAR,4
```

plot load-lateral displacement curve

```
VAR,5
AXLAB,X,l-DISPLACEMENT
AXLAB,Y,LOAD
LVAR,4
```

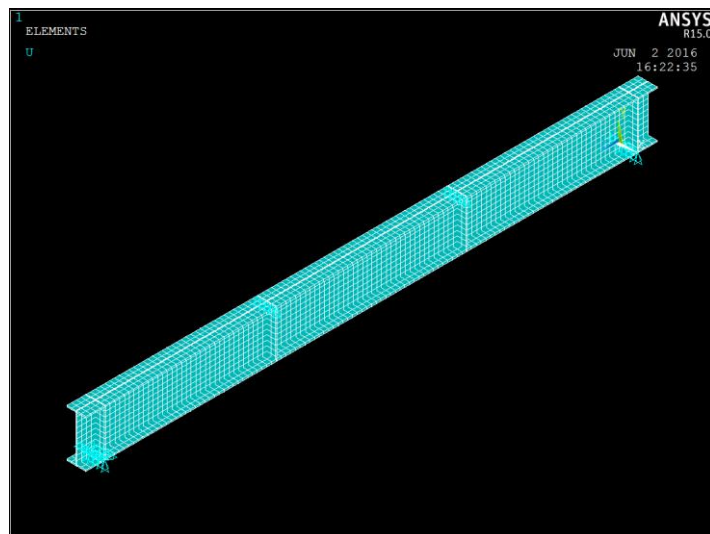
output data to txt

```
OUTPUT,RESTDATA,TXT
RVAR,2,4,5
OUTPUT
```

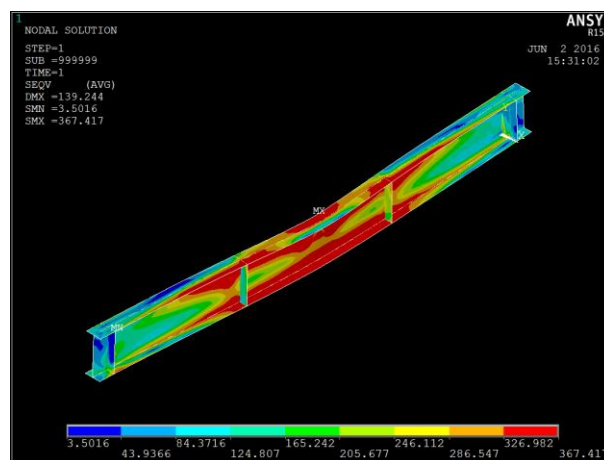
INISH

后处理

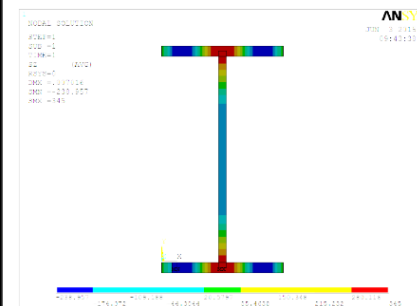
2.3 APDL参数化建模



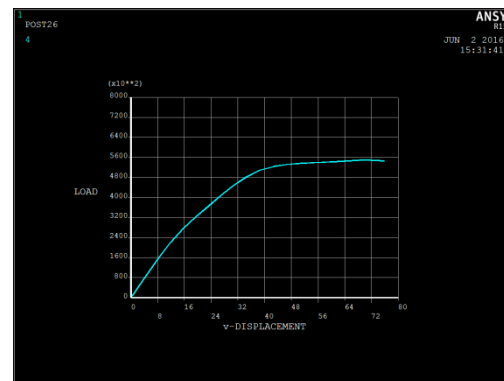
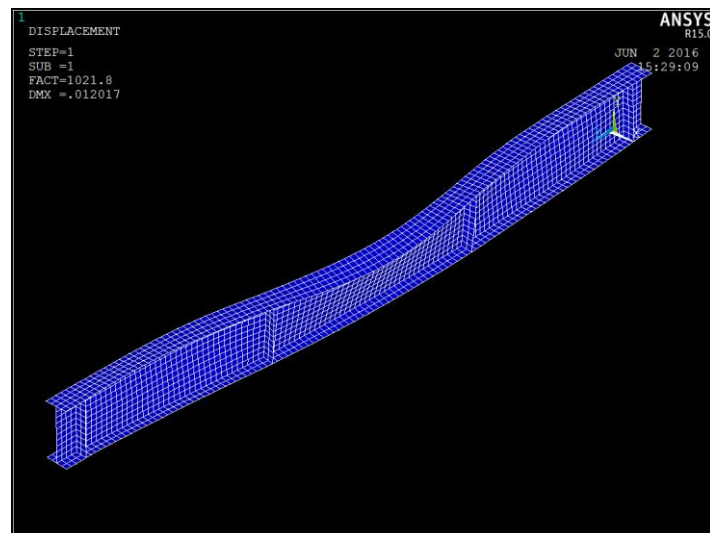
模型



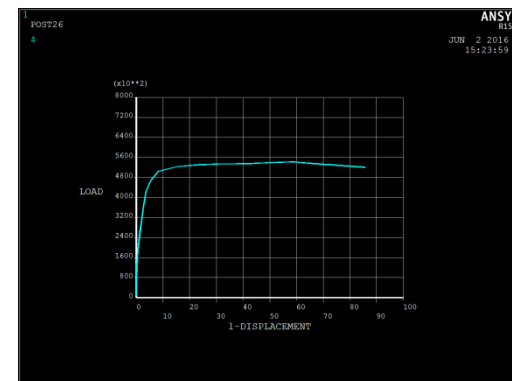
应力云图



残余应力分布



荷载跨中位移曲线
一阶屈曲变形



荷载侧向位移曲线



2.4 Python参数化建模

```
1 #Section Dimension Coordinate
2 LG=200
3 LK=500
4 X_HalfFlange=100.000
5 Y_HalfWeb=238.000
6 tf=24.000
7 tw=12.000
8 Offset_FlangeWeld=8.000
9 Offset_WebWeld=230.00017:29:44
10 AxialMeshDimension=20
11 d_web= Offset_WebWeld/10
12 d_flange=(X_HalfFlange-Offset_FlangeWeld)/10
13
14 L=5800
15 F=115e6/(0.25*L)
16
17 # Model Name
18 LG=str(LG)
19 LK=str(LK)
20 LC=str(L)
21 PH='Q345'
22 ModelName='I'+ '-' +LG+'-' +LK+'-' +LC+'-' +PH+'-JZL'
23
24 #Residual Stress
25 F01= 271.71
26 F02= 125.13
27 F03= -21.46
28 F04= -142.16
29 F05= -189.58
30 F06= -189.58
31 F07= -189.58
32 F08= -189.58
33 F09= -69.79
34 F10= 50.00
35 FW= 345.00
36 W01= -116.13
37 W02= -116.13
38 W03= -116.13
39 W04= -116.13
40 W05= -116.13
41 W06= -91.08
42 W07= -14.65
43 W08= 88.11
44 W09= 190.86
45 W10=293.62
46
47 #Imperfection
48 order=1
49 amp=L/1000.0
50
```

几何尺寸
草图坐标

模型命名

截面残余应力
大小定义

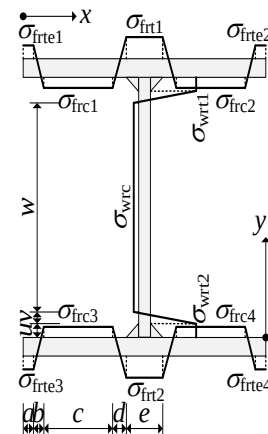
```
51 from part import *
52 from material import *
53 from section import *
54 from assembly import *
55 from step import *
56 from interaction import *
57 from load import *
58 from mesh import *
59 from optimization import *
60 from job import *
61 from sketch import *
62 from visualization import *
63 from connectorBehavior import *
64 from abaqusConstants import *
65 import string
```

导入模块

EditPlus.exe 文本编辑器
.py

2.4 Python参数化建模

翼缘宽 B/mm	翼缘厚 t_f/mm	梁高 H/mm	腹板厚 t_w/mm	焊脚 h_f/mm	钢材牌号	屈服强度 f_y/MPa		翼缘外伸宽 b_f/mm	腹板高 h_0/mm
200.000	24.000	500.000	12.000	8	Q460	460		94.000	452.000
a/mm	b/mm	c/mm	d/mm	e/mm	u/mm	v/mm	w/mm	翼缘焊缝坐标 $/mm$	腹板焊缝坐标
8.600	8.600	37.436	31.364	28.000	8.000	97.829	240.343	14.000	218.000
σ_{ft}/MPa	σ_{fte}/MPa	σ_{fc}/MPa	σ_{wft}/MPa	σ_{wfc}/MPa		翼缘单元宽	腹板	单元最大长度 $/mm$	
345.00	50.00	-189.58	345.00	-116.13		8.600	21.800	25.800	
x/mm	$\Delta x/mm$	σ/MPa	x/mm	$\Delta x/mm$	位置	σ/MPa	编号	σ/MPa	
0.000		345.00	0.000	0.000	受拉	345.00		345.00	
14.000	14.000	345.00	14.000	14.000	受拉	345.00		345.00	
45.364	31.364	-189.58	22.600	8.600	过渡区1	198.42	F01	271.71	
82.800	37.436	-189.58	31.200	8.600	过渡区1	51.84	F02	125.13	
91.400	8.600	50.00	39.800	8.600	过渡区1	-94.75	F03	-21.46	
100.000	8.600	50.00	48.400	8.600	受压	-189.58	F04	-142.16	
			57.000	8.600	受压	-189.58	F05	-189.58	
			65.600	8.600	受压	-189.58	F06	-189.58	
			74.200	8.600	受压	-189.58	F07	-189.58	
			82.800	8.600	受压	-189.58	F08	-189.58	
			91.400	8.600	过渡区2	50.00	F09	-69.79	
			100.000	8.600	边缘受拉	50.00	F10	50.00	
y/mm	$\Delta y/mm$	σ/MPa	y/mm	$\Delta y/mm$	位置	σ/MPa	编号	σ/MPa	
0.000		-116.13	0.000	0.000	受压	-116.13		-116.13	
120.171	120.171	-116.13	21.800	21.800	受压	-116.13	W01	-116.13	
218.000	97.829	345.00	43.600	21.800	受压	-116.13	W02	-116.13	
238.000	20.000	345.00	65.400	21.800	受压	-116.13	W03	-116.13	
			87.200	21.800	受压	-116.13	W04	-116.13	
			109.000	21.800	受压	-116.13	W05	-116.13	
			130.800	21.800	过渡区	-66.03	W06	-91.08	
			152.600	21.800	过渡区	36.73	W07	-14.65	
			174.400	21.800	过渡区	139.49	W08	88.11	
			196.200	21.800	过渡区	242.24	W09	190.86	
			218.000	21.800	过渡区	345.00	W10	293.62	
			238.000	20.000	受拉	345.00		345.00	



残余应力分布



2.4 Python参数化建模

```
82 # Create a new model
83 mdb.Model(modelType=STANDARD_EXPLICIT, name=ModelName)
84 MyModel=mdb.models[ModelName]
85
86 # 1.Create new parts
87 PartName=ModelName
88 MyModel.ConstrainedSketch(name='profile', sheetSize=1000.0)
89 MyProfile=MyModel.sketches['profile']
90 MyProfile.Line (point1=(-X_HalfFlange, Y_HalfWeb), point2=(X_HalfFlange, Y_HalfWeb))
91 MyProfile.Line (point1=(-X_HalfFlange, -Y_HalfWeb), point2=(X_HalfFlange, -Y_HalfWeb))
92 MyProfile.Line (point1=(0.0, Y_HalfWeb), point2=(0.0, -Y_HalfWeb))
93 MyModel.Part(dimensionality=THREE_D, name=PartName, type=DEFORMABLE_BODY)
94 MyPart=MyModel.parts[PartName]
95 MyPart.BaseShellExtrude(sketch=MyProfile, depth=L)
96 del MyModel.sketches['profile']
97
98 PartName='EndPlatePositive'
99 MyModel.ConstrainedSketch(name='profile', sheetSize=1000.0)
100 MyProfile=MyModel.sketches['profile']
101 MyProfile.rectangle(point1=(X_HalfFlange, Y_HalfWeb), point2=(0.0, -Y_HalfWeb))
102 MyModel.Part(dimensionality=THREE_D, name=PartName, type=DEFORMABLE_BODY)
103 MyPart_EPP=MyModel.parts[PartName]
104 MyPart_EPP.BaseShell(sketch=MyProfile)
105 del MyModel.sketches['profile']
106
107 PartName='EndPlateNegative'
108 MyModel.ConstrainedSketch(name='profile', sheetSize=1000.0)
109 MyProfile=MyModel.sketches['profile']
110 MyProfile.rectangle(point1=(-X_HalfFlange, Y_HalfWeb), point2=(0.0, -Y_HalfWeb))
111 MyModel.Part(dimensionality=THREE_D, name=PartName, type=DEFORMABLE_BODY)
112 MyPart_EPN=MyModel.parts[PartName]
113 MyPart_EPN.BaseShell(sketch=MyProfile)
114 del MyModel.sketches['profile']
115
116 # 2.Define material and sectional property
117
118 #Q345
119 MyModel.Material(name='Q345')
120 MyModel.materials['Q345'].Elastic(table=((206000.0,
121     0.3), ))
122 MyModel.materials['Q345'].Plastic(table=((345, 0.0),
123     (351.900, 0.01809), (576.0, 0.17953)))
124 ...
```

定义材料

名义应力Vs真实应力

名义应变Vs真实应变Vs塑性应变

创建Part

2.4 Python参数化建模

```
161 #Section Thickness
162 MyModel.HomogeneousShellSection(name='flange',
163     preIntegrate=OFF, material=PH, thicknessType=UNIFORM, thickness=tf,
164     thicknessField='', idealization=NO_IDEALIZATION, poissonDefinition=DEFAULT,
165     thicknessModulus=None, temperature=GRADIENT, useDensity=OFF,
166     integrationRule=SIMPSON, numIntPts=5)
167 MyModel.HomogeneousShellSection(name='web',
168     preIntegrate=OFF, material=PH, thicknessType=UNIFORM, thickness=tw,
169     thicknessField='', idealization=NO_IDEALIZATION, poissonDefinition=DEFAULT,
170     thicknessModulus=None, temperature=GRADIENT, useDensity=OFF,
171     integrationRule=SIMPSON, numIntPts=5)
172 MyModel.HomogeneousShellSection(name='EndPlate',
173     preIntegrate=OFF, material=PH, thicknessType=UNIFORM, thickness=tf,
174     thicknessField='', idealization=NO_IDEALIZATION, poissonDefinition=DEFAULT,
175     thicknessModulus=None, temperature=GRADIENT, useDensity=OFF,
176     integrationRule=SIMPSON, numIntPts=5)
177 #Datum
178 MyPart.DatumPlaneByPrincipalPlane(principalPlane=YZPLANE, offset=Offset_FlangeWeld)
179 MyPart.DatumPlaneByPrincipalPlane(principalPlane=YZPLANE, offset=-Offset_FlangeWeld)
180 MyPart.DatumPlaneByPrincipalPlane(principalPlane=XZPLANE, offset=Offset_WebWeld)
181 MyPart.DatumPlaneByPrincipalPlane(principalPlane=XZPLANE, offset=-Offset_WebWeld)
182 MyPart.DatumPlaneByPrincipalPlane(principalPlane=XYPLANE, offset=L/2)
183 MyPart.DatumCsysByThreePoints(name='flange', coordSysType=CARTESIAN, origin=(0.0,
184     0.0, 0.0), point1=(0.0, 0.0, 1000.0), point2=(1000.0, 0.0, 0.0))
185 MyPart.DatumCsysByThreePoints(name='web', coordSysType=CARTESIAN, origin=(0.0, 0.0,
186     0.0), point1=(0.0, 0.0, 1000.0), point2=(0.0, 1000.0, 0.0))
187 [n1,n2,n3,n4,n5,n6,n7]= MyPart.datums.keys()
188 #Partition Face
189 MyPart.PartitionFaceByDatumPlane(datumPlane=MyPart.datums[n1], faces=MyPart.faces)
190 MyPart.PartitionFaceByDatumPlane(datumPlane=MyPart.datums[n2], faces=MyPart.faces)
191 MyPart.PartitionFaceByDatumPlane(datumPlane=MyPart.datums[n3], faces=MyPart.faces)
192 MyPart.PartitionFaceByDatumPlane(datumPlane=MyPart.datums[n4], faces=MyPart.faces)
193 MyPart.PartitionFaceByDatumPlane(datumPlane=MyPart.datums[n5], faces=MyPart.faces)
```

板件厚度定义

定义基准面

定义剖分面



2.4 Python参数化建模

```
204 # Assign Section
205 faces=MyPart.faces.findAt(((Offset_FlangeWeld/2, Y_HalfWeb, L/4), ), ((-Offset_Flange
206 MyPart.Set(faces=faces, name='Flange')
207 faces=MyPart.faces.findAt(((0.0, 0.0, L/4), ), ((0.0, 0.0, 3*L/4), ), ((0.0, (Offset_
208 MyPart.Set(faces=faces, name='Web')
209 faces=MyPart_EPP.faces
210 MyPart_EPP.Set(faces=faces, name='EPP')
211 faces=MyPart_EPN.faces
212 MyPart_EPN.Set(faces=faces, name='EPN')
213
214 region = MyPart.sets['Flange']
215 MyPart.SectionAssignment(region=region, sectionName='flange', offset=0.0,
216     offsetType=MIDDLE_SURFACE, offsetField='',
217     thicknessAssignment=FROM_SECTION)
218 region = MyPart.sets['Web']
219 MyPart.SectionAssignment(region=region, sectionName='web', offset=0.0,
220     offsetType=MIDDLE_SURFACE, offsetField='',
221     thicknessAssignment=FROM_SECTION)
222 region = MyPart_EPP.sets['EPP']
223 MyPart_EPP.SectionAssignment(region=region, sectionName='EndPlate', offset=0.0,
224     offsetType=MIDDLE_SURFACE, offsetField='',
225     thicknessAssignment=FROM_SECTION)
226 region = MyPart_EPN.sets['EPN']
227 MyPart_EPN.SectionAssignment(region=region, sectionName='EndPlate', offset=0.0,
228     offsetType=MIDDLE_SURFACE, offsetField='',
229     thicknessAssignment=FROM_SECTION)
230
231 region = MyPart.sets['Flange']
232 orientation = MyPart.datums[n6]
233 MyPart.MaterialOrientation(
234     region=region, orientationType=SYSTEM, axis=AXIS_3, localCsys=orientation,
235     fieldName='', additionalRotationType=ROTATION_NONE, angle=0.0,
236     additionalRotationField='')
237 region = MyPart.sets['Web']
238 orientation = MyPart.datums[n7]
239 MyPart.MaterialOrientation(
240     region=region, orientationType=SYSTEM, axis=AXIS_3, localCsys=orientation,
241     fieldName='', additionalRotationType=ROTATION_NONE, angle=0.0,
242     additionalRotationField='')
```

赋予板件厚度

定义板件局部方向



2.4 Python参数化建模

装配构件

创建特征集合

定义初始荷载步

建立参考点与
特征集合之间的联系

```
244 # 3.Create assembly
245 # Create assembly
246 MyAssembly = MyModel.rootAssembly
247 MyAssembly.DatumCsysByDefault(CARTESIAN)
248 MyAssembly.Instance(name=ModelName, part=MyPart, dependent=ON)
249 MyInstance=MyAssembly.instances[ModelName]
250 MyAssembly.Instance(name='EndPlatePositive', part=MyPart_EPP, dependent=ON)
251 MyInstance_EPP=MyAssembly.instances['EndPlatePositive']
252 MyAssembly.Instance(name='EndPlateNegative', part=MyPart_EPN, dependent=ON)
253 MyInstance_EPN=MyAssembly.instances['EndPlateNegative']
254 # Create Feature
255 edges=MyInstance.edges.findAt(((0.0, Offset_WebWeld/2, 0.0), ), ((0.0, -Offset_WebWeld/2, 0.0), ), ((0.0, (Offset_WebWeld+ Y_HalfWe
256 MyAssembly.Set(edges=edges, name='EndEdge1')
257 edges=MyInstance.edges.findAt(((0.0, Offset_WebWeld/2, L), ), ((0.0, -Offset_WebWeld/2, L), ), ((0.0, (Offset_WebWeld+ Y_HalfWeb)/2
258 MyAssembly.Set(edges=edges, name='EndEdge2')
259
260
261 MyAssembly.ReferencePoint(point=(0.0, 0.0, 0.0))
262 MyAssembly.ReferencePoint(point=(0.0, 0.0, L))
263 [m1,m2]= MyAssembly.referencePoints.keys()
264 MyAssembly.Set(referencePoints= (MyAssembly.referencePoints[m2],), name='RP1')
265 MyAssembly.Set(referencePoints= (MyAssembly.referencePoints[m1],), name='RP2')
266
267 # 4.Create step
268 #Step definition
269 MyModel.BuckleStep(name='Step-1', previous='Initial',
270     numEigen=1, vectors=2)
271
272
273 # 5.Interaction
274 region1=MyAssembly.sets['RP1']
275 region2= MyAssembly.sets['EndEdge1']
276 MyModel.Coupling(name='End1', controlPoint=region1,
277     surface=region2, influenceRadius=WHOLE_SURFACE, couplingType=KINEMATIC,
278     localCsys=None, u1=ON, u2=ON, u3=ON, ur1=ON, ur2=ON, ur3=ON)
279 region1=MyAssembly.sets['RP2']
280 region2= MyAssembly.sets['EndEdge2']
281 MyModel.Coupling(name='End2', controlPoint=region1,
282     surface=region2, influenceRadius=WHOLE_SURFACE, couplingType=KINEMATIC,
283     localCsys=None, u1=ON, u2=ON, u3=ON, ur1=ON, ur2=ON, ur3=ON)
```



2.4 Python参数化建模

```
373 # 6.Load
374 region = MyAssembly.sets['RP1']
375 MyModel.DisplacementBC(name='End1',
376     createStepName='Initial', region=region, u1=SET, u2=SET, u3=SET, url=UNSET,
377     ur2=UNSET, ur3=SET, amplitude=UNSET, distributionType=UNIFORM,
378     fieldName='', localCsys=None)
379 region = MyAssembly.sets['RP2']
380 MyModel.DisplacementBC(name='End2',
381     createStepName='Initial', region=region, u1=SET, u2=SET, u3=UNSET, url=UNSET,
382     ur2=UNSET, ur3=SET, amplitude=UNSET, distributionType=UNIFORM,
383     fieldName='', localCsys=None)
384
385 region = MyAssembly.sets['RP2']
386 MyModel.ConcentratedForce(name='End2_Load', createStepName='Step-1',
387     region=region, cf3=-1.0, distributionType=UNIFORM, field='',
388     localCsys=None)
389
390 # 7.Create mesh
391 # Mesh control
392 pickedRegions = MyPart.faces
393 MyPart.setMeshControls(regions=pickedRegions, technique=SWEEP)
394
395 # Seed
396 MyPart.seedPart(size=AxialMeshDimension, deviationFactor=0.1, minSizeFactor=0.1)
397
435 #Mesh
436 MyPart.generateMesh()
437 MyPart_EPP.generateMesh()
438 MyPart_EPN.generateMesh()
439
440 elements=MyPart.elements.getByBoundingBox(yMin=-d_web-1, yMax=d_web+1)
441 MyPart.Set(elements=elements, name='W01')
442
530 # 8.Job
531 JobName=ModelName+'-Buckle'
532
533 mdb.Job(name= JobName, model= ModelName, description='',
534     type=ANALYSIS, atTime=None, waitMinutes=0, waitHours=0, queue=None,
535     memory=90, memoryUnits=PERCENTAGE, getMemoryFromAnalysis=True,
536     explicitPrecision=SINGLE, nodalOutputPrecision=SINGLE, echoPrint=OFF,
537     modelPrint=OFF, contactPrint=OFF, historyPrint=OFF, userSubroutine='',
538     scratch='', multiprocessingMode=DEFAULT, numCpus=1, numGPUs=0)
```

施加荷载及约束

划分网格
S4R单元

提交屈曲分析任务



2.4 Python参数化建模

```
543 # 9.Update Step
544 #Step definiton
545 regionDef=MyAssembly.sets['MPU']
546 MyModel.StaticRiksStep(name='Step-1',
547     previous='Initial', maintainAttributes=True, nodeOn=ON,
548     maximumDisplacement=100.0, region=regionDef, dof=1, initialArcInc=0.05,
549     nlgeom=ON)

551 # Create history output
552 regionDef= MyAssembly.sets['MPU']
553 MyModel.HistoryOutputRequest(name='H-Output-2',
554     createStepName='Step-1', variables=('U1', 'U2'), region=regionDef,
555     sectionPoints=DEFAULT, rebar=EXCLUDE)
556 regionDef= MyAssembly.sets['MPD']
557 MyModel.HistoryOutputRequest(name='H-Output-3',
558     createStepName='Step-1', variables=('U1', 'U2'), region=regionDef,
559     sectionPoints=DEFAULT, rebar=EXCLUDE)
560 regionDef= MyAssembly.sets['MPM']
561 MyModel.HistoryOutputRequest(name='H-Output-4',
562     createStepName='Step-1', variables=('U1', 'U2'), region=regionDef,
563     sectionPoints=DEFAULT, rebar=EXCLUDE)

565 # 10.Update Load
566 MyModel.constraints['End1'].suppress()
567 MyModel.constraints['End2'].suppress()
568 MyModel.loads['End2_Load'].suppress()
569 MyModel.boundaryConditions['End1'].suppress()
570 MyModel.boundaryConditions['End2'].suppress()
571
572 region = MyAssembly.sets['EndEdge1']
573 MyModel.DisplacementBC(name='EndEdge1',
574     createStepName='Initial', region=region, u1=SET, u2=SET, u3=UNSET, ur1=UNSET,
575     ur2=UNSET, ur3=UNSET, amplitude=UNSET, distributionType=UNIFORM,
576     fieldName='', localCsys=None)
577 region = MyAssembly.sets['EndEdge2']
578 MyModel.DisplacementBC(name='EndEdge2',
579     createStepName='Initial', region=region, u1=SET, u2=SET, u3=UNSET, ur1=UNSET,
580     ur2=UNSET, ur3=UNSET, amplitude=UNSET, distributionType=UNIFORM,
581     fieldName='', localCsys=None)
```

更新荷载步

产生历史输出数据

更新荷载及约束



2.4 Python参数化建模

```
593 # 11.Residual stress & Imperfection
594 # Residual stress
595 region = MyInstance.sets['F01']
596 MyModel.Stress(name='F01', region=region,
597     distributionType=UNIFORM, sigma11=F01, sigma22=0.0, sigma12=0.0,
598     sigma33=None, sigma13=None, sigma23=None)
599 region = MyInstance.sets['F02']
600 MyModel.Stress(name='F02', region=region,
601     distributionType=UNIFORM, sigma11=F02, sigma22=0.0, sigma12=0.0,
602     sigma33=None, sigma13=None, sigma23=None)
603 region = MyInstance.sets['F03']
604 MyModel.Stress(name='F03', region=region,
605     distributionType=UNIFORM, sigma11=F03, sigma22=0.0, sigma12=0.0,
606     sigma33=None, sigma13=None, sigma23=None)
607 region = MyInstance.sets['F04']
608 MyModel.Stress(name='F04', region=region,
609     distributionType=UNIFORM, sigma11=F04, sigma22=0.0, sigma12=0.0,
610     sigma33=None, sigma13=None, sigma23=None)

680 Place='*Initial Conditions, type=STRESS\n'+ModelName+'.W10, '+str(W10)+' , 0., 0.,
681 MyModel.keywordBlock.synchVersions(storeNodesAndElements=False)
682 position1 = GetKeywordPosition(ModelName, Place)
683 MyModel.keywordBlock.insert(position1, '\n*Initial Conditions, type=STRESS,\n'+unbalanced stress=step')
685 # Imperfection
686 imper=str(order)+' ,'+str(amp)+'\n'
687
688 MyModel.keywordBlock.synchVersions(storeNodesAndElements=False)
689 position1 = GetKeywordPosition(ModelName, '*End Assembly')
690 MyModel.keywordBlock.insert(position1, '\n*Imperfection, file='+JobName+', step=1\n'+imper)
692 # 12.Job
693 mdb.Job(name= ModelName, model= ModelName, description='',
694     type=ANALYSIS, atTime=None, waitMinutes=0, waitHours=0, queue=None,
695     memory=90, memoryUnits=PERCENTAGE, getMemoryFromAnalysis=True,
696     explicitPrecision=SINGLE, nodalOutputPrecision=SINGLE, echoPrint=OFF,
697     modelPrint=OFF, contactPrint=OFF, historyPrint=OFF, userSubroutine='',
698     scratch='', multiprocessingMode=DEFAULT, numCpus=8, numDomains=8,
699     numGPUs=0)
700 mdb.jobs[ModelName].submit()
```

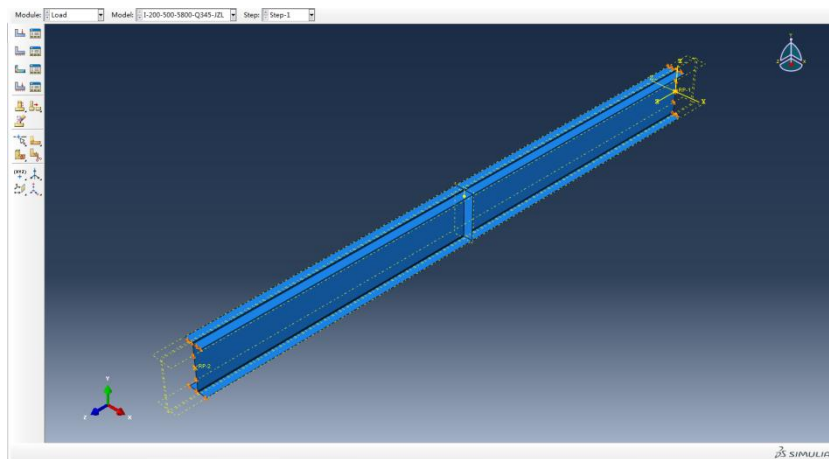
输入残余应力

使残余应力满足自平衡

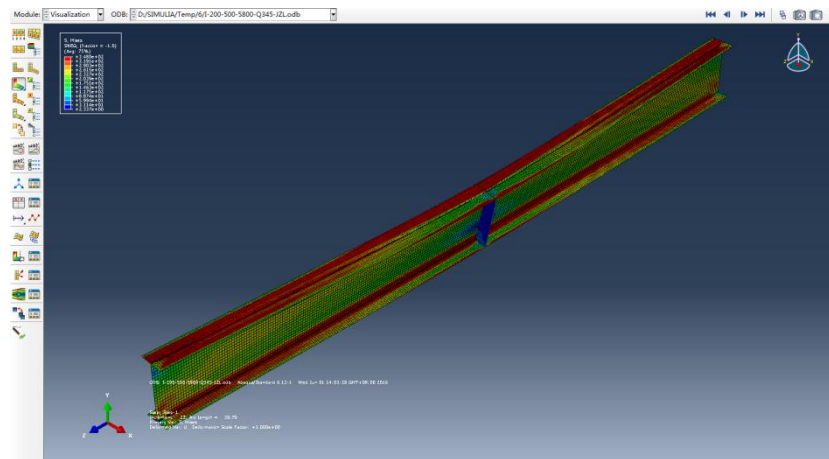
输入几何初始缺陷

提交任务

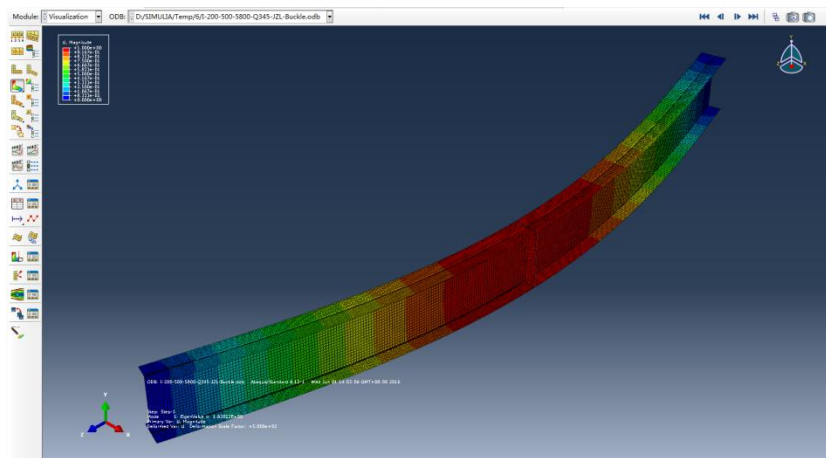
2.4 Python参数化建模



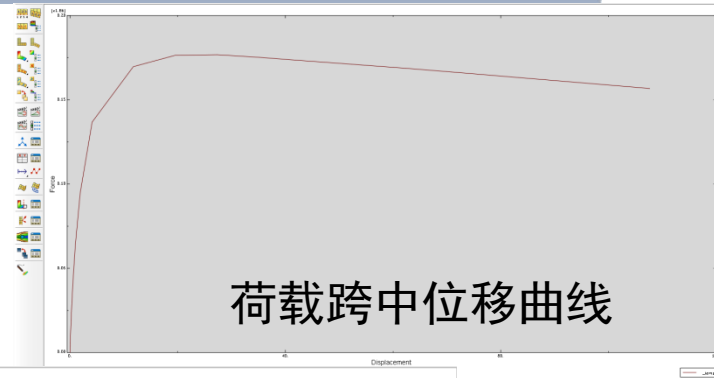
模型



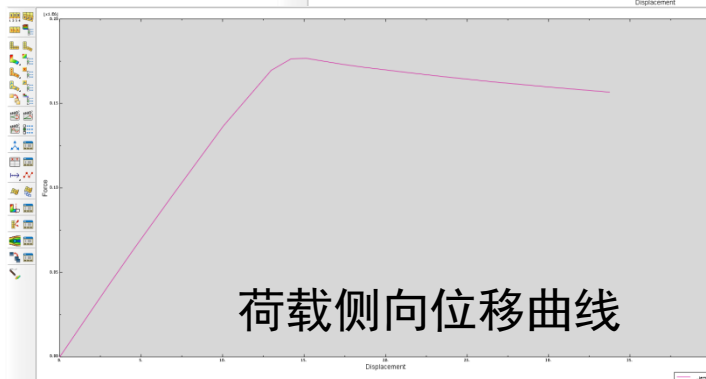
应力云图



一阶屈曲变形



荷载跨中位移曲线



荷载侧向位移曲线



◎ 3. 课题结论

- 对比分析APDL与Python建模过程及模型要素

- 几何建模
- 材料模型
- 单元类型
- 收敛准则

- 对比分析APDL与Python分析结果

- 结果误差
- 建模复杂程度
- 收敛难易
- 计算效率

感悟：

- 打好基础
- 深入理解
- 多学多练

谢谢大家!

