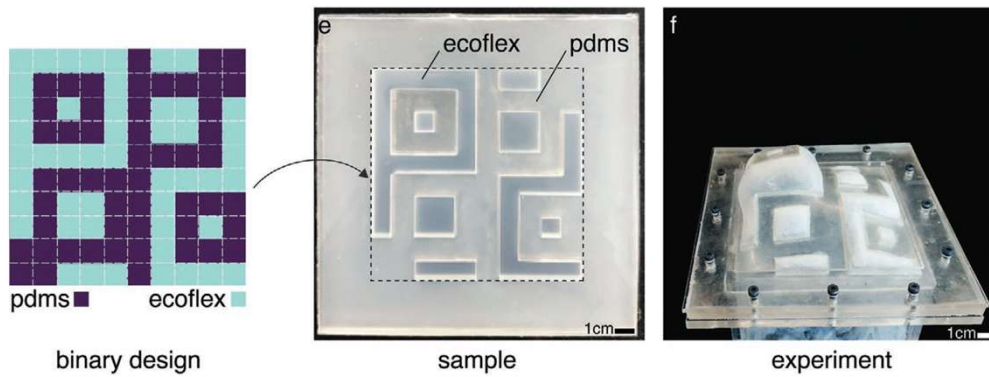
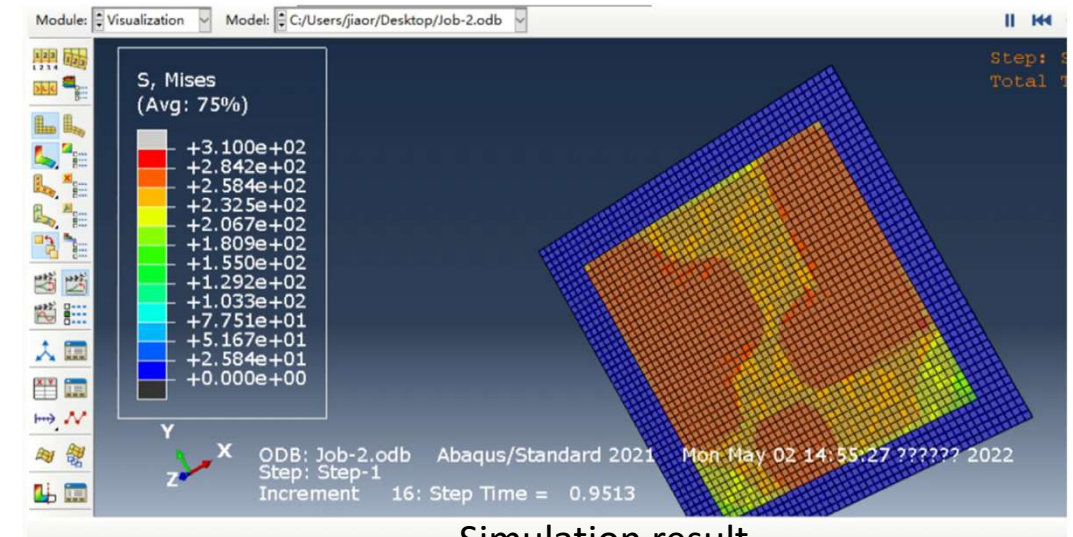
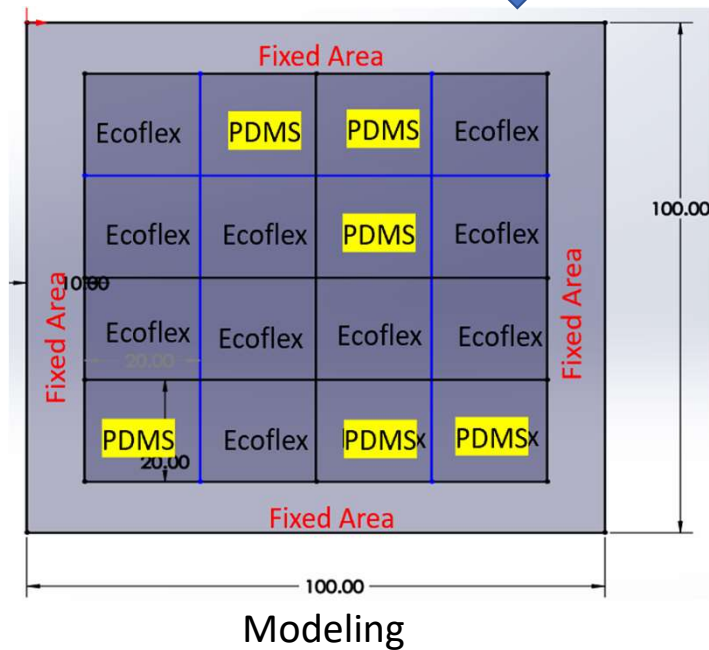


Abaqus Tutorial:  
PDMS + Ecoflex 0030 binary material membrane  
design, modeling, simulation

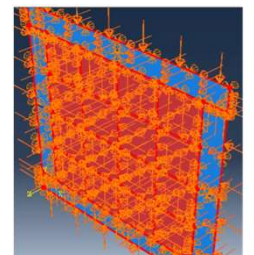
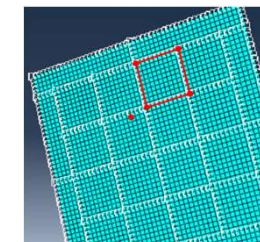
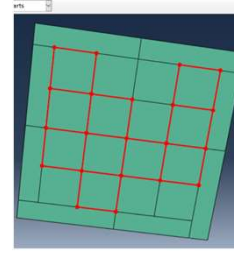
JRW  
2022/05/02



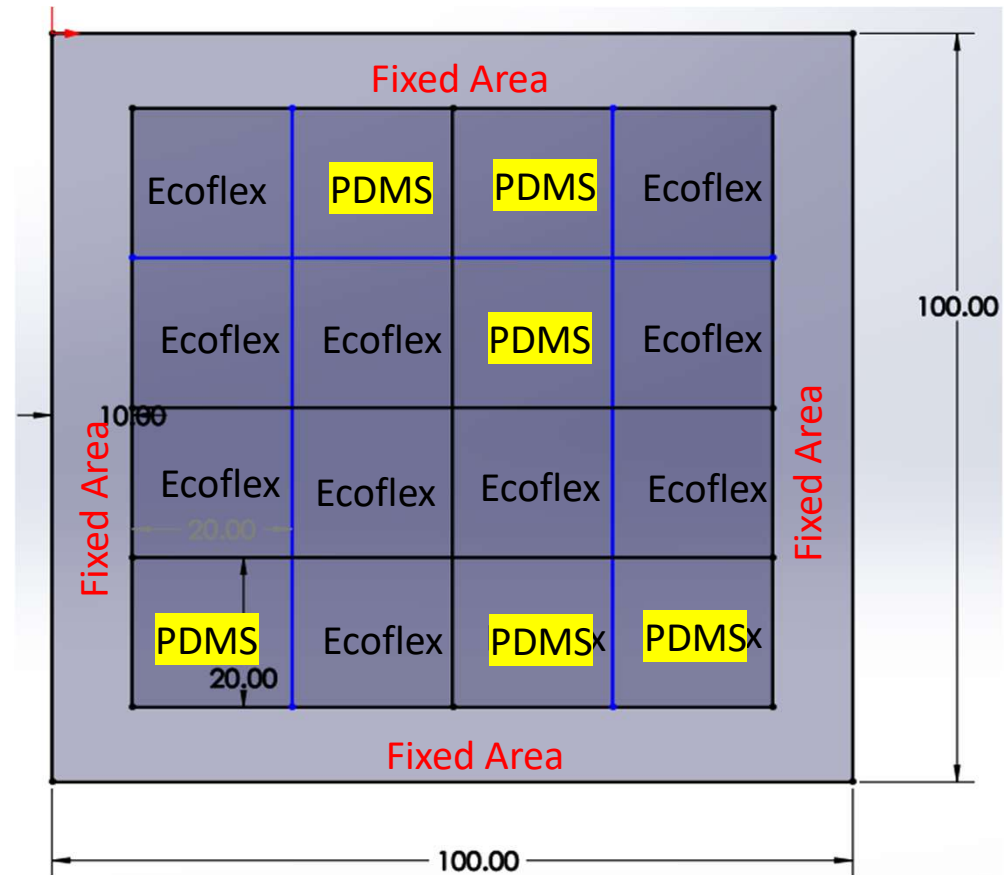
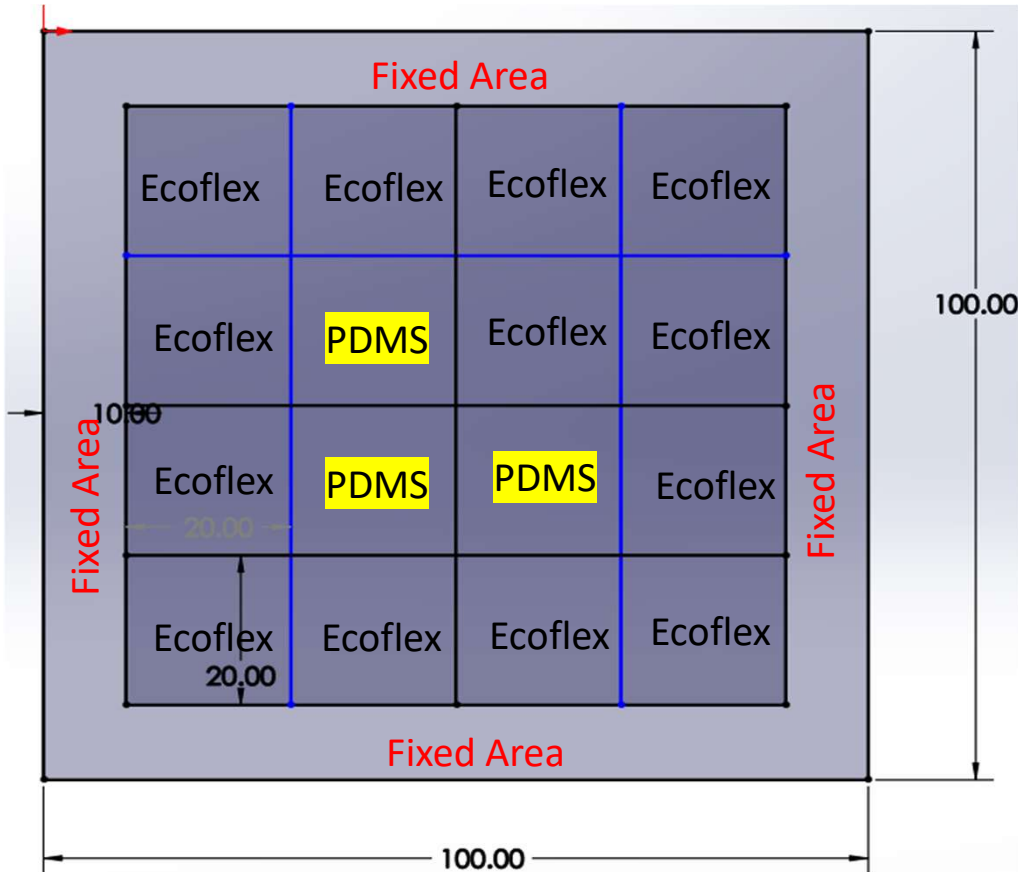
Paper refer to: bi-material pixel membrane design  
Antonio Elia Forte, 2022



Abaqus Setting	
Material	PDMS: 500 kPa/0.4 Ecoflex 0030: 29.5 kPa and 0.45
Load Parameter	Pressure: 500 Pa (Centered Area); Boundary condition: brim fixed
Modeling:	Shell Thickness: 0.2 (Unit: mm)
Mesh Size:	2mm



# Design 2D model



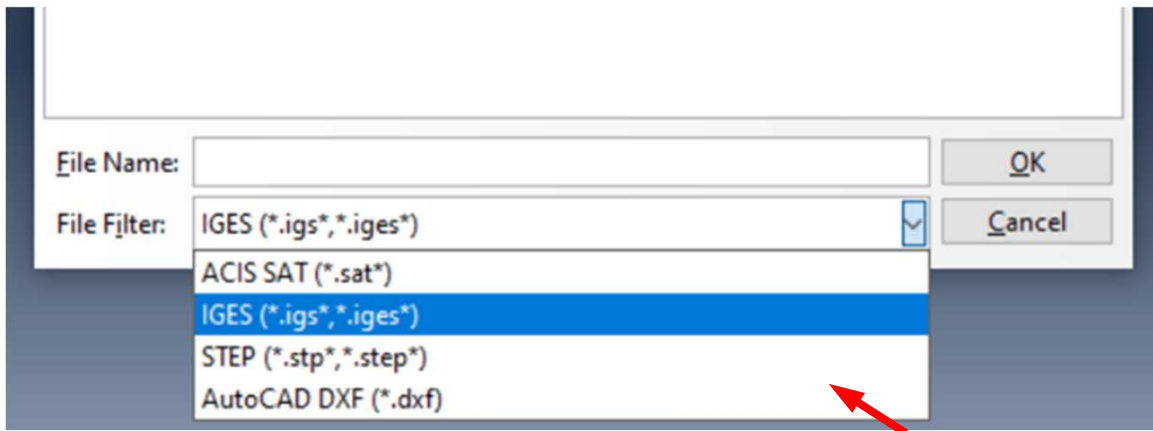
## Abaqus simulation setting

- Modeling: Shell Thickness: 0.2 (Unit: mm)
- Load Parameter setting: Pressure-500 Pa (Centered Area); Boundary condition-brim fixed
- Material:

PDMS: Young's modulus of 500 kPa + Poisson's ratio of 0.4  
 Ecoflex 0030: for the 1:1:0 mixture proportions 29.5 kPa and 0.45,

# Import 2D patterns

- Click File > Import > Sketch
- Choose the directory, then select “IGES” in File Filter
- Select the .iges file that you want to import, click “OK”
- Click “OK” in the pop-up window



# Create part from sketch

- Click Module > Part > create part > Shape > Shell > Type > Planar > Continue (Fig.1)
- Click Add Sketch (Fig.2), select the sketch you just added, click “OK”
- Click “Done” (Fig.3), or click mouse wheel
- Click “Done” again (Fig.4), or click mouse wheel

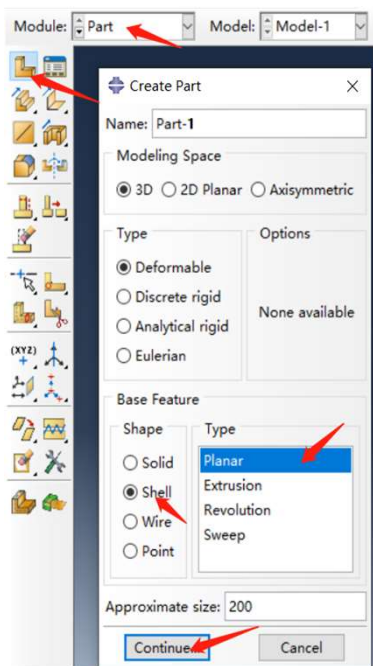


Figure 1

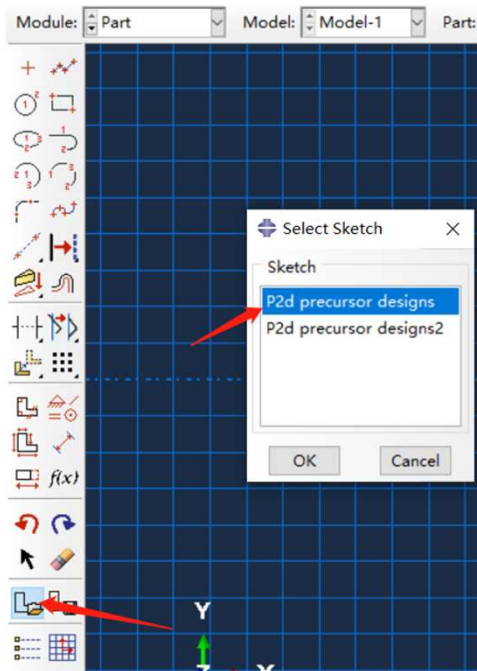


Figure 2



Figure 3

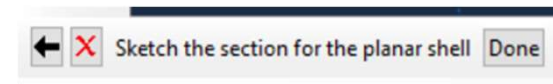
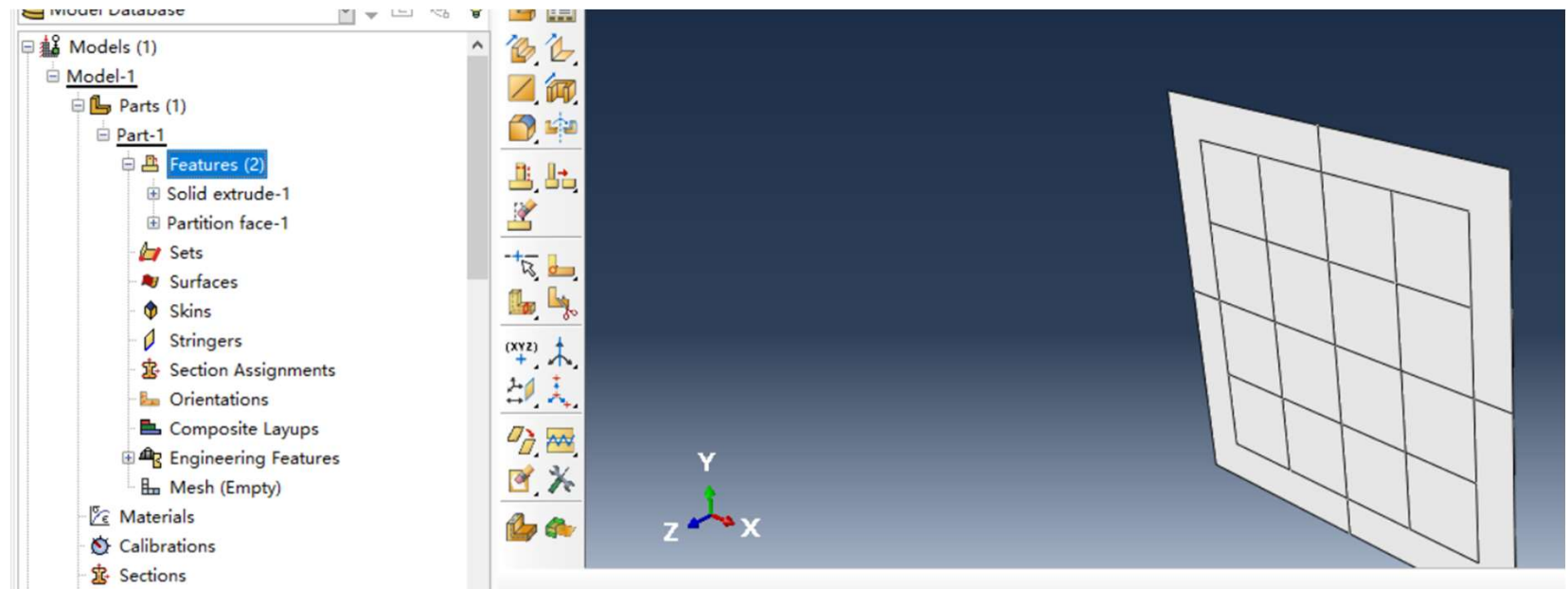
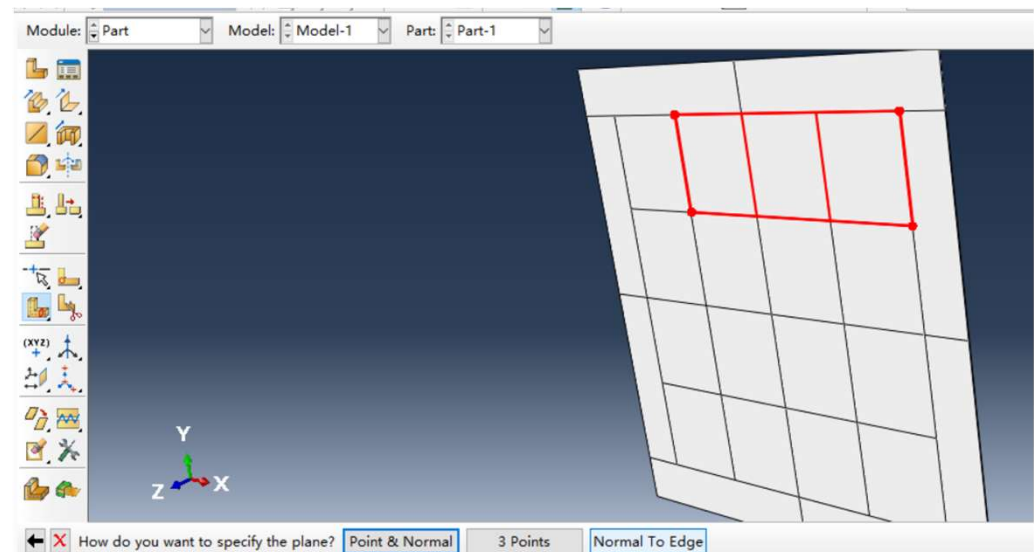


Figure 4

# Partition Part



- Click “Partition Face: Sketch”
- Click an edge of the part, shows up the sketch window
- Then cut it into partition cell



# Create Material

- Click Module > Property > Create Material (Fig.1)
- In the Edit Material window, create a name for this material (Fig.2)
- Click Mechanical > Elasticity > Elastic (Fig.2)
- Input the Young's Modulus and Poisson's Ratio (500000 Pa and 0.5 for PDMS)(Fig.3)

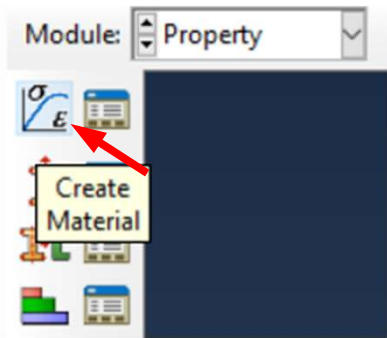


Figure 1

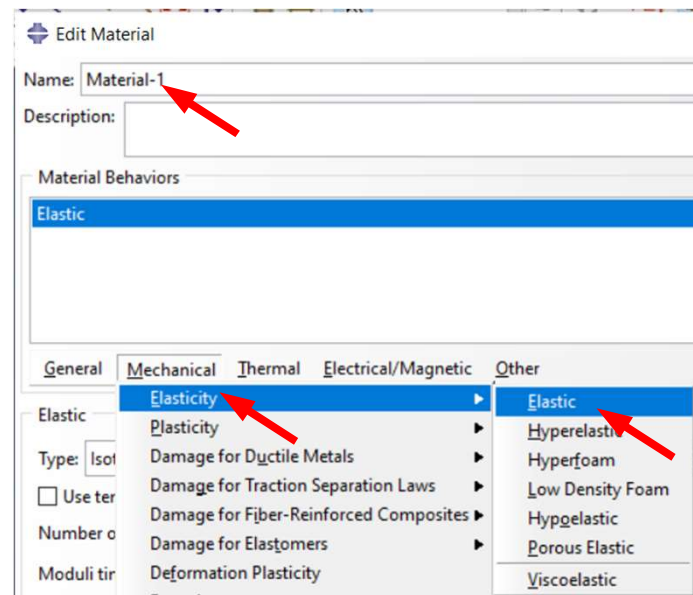


Figure 2

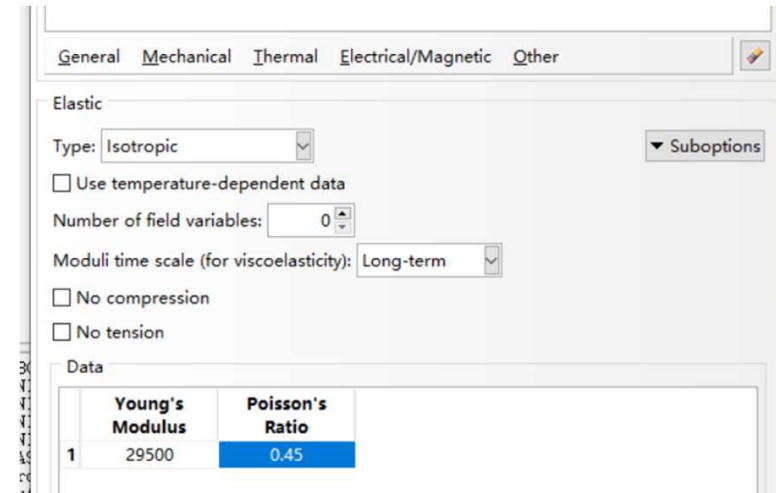


Figure 3



# Create Section

- In Property Module, click Create Section (Fig.1)
- In Create Section window, create a name for this section (Fig.1)
- Click Shell > Homogeneous > Continue (Fig.1)
- In Edit Section window, set the shell thickness as ~~0.003~~, select the material you just created (Fig.2)
- Click “OK” (Fig.2)

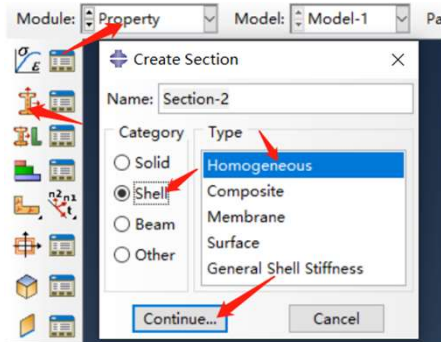


Figure 1

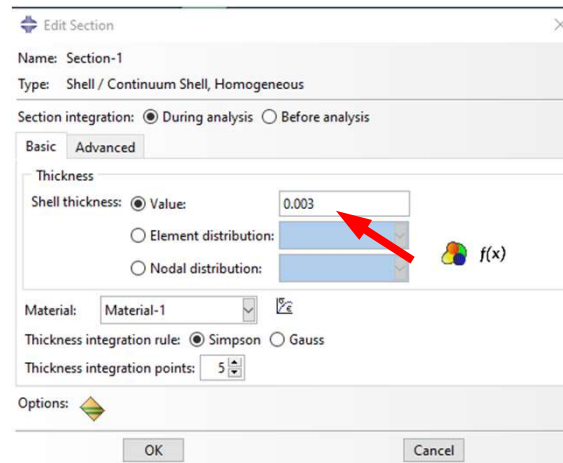
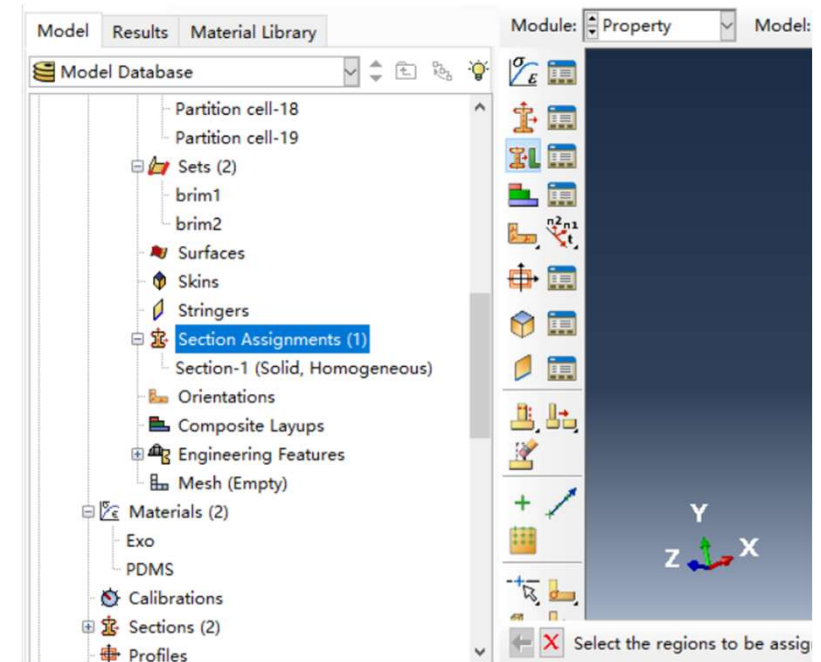


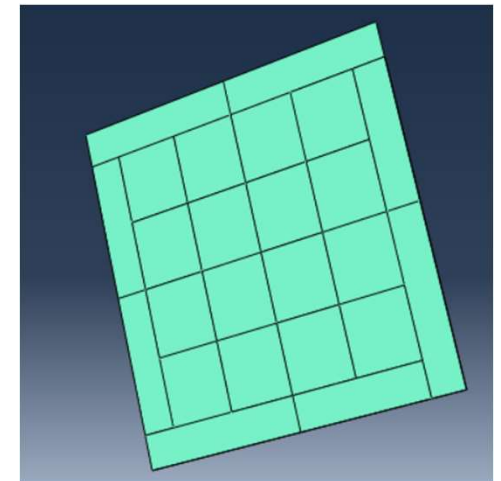
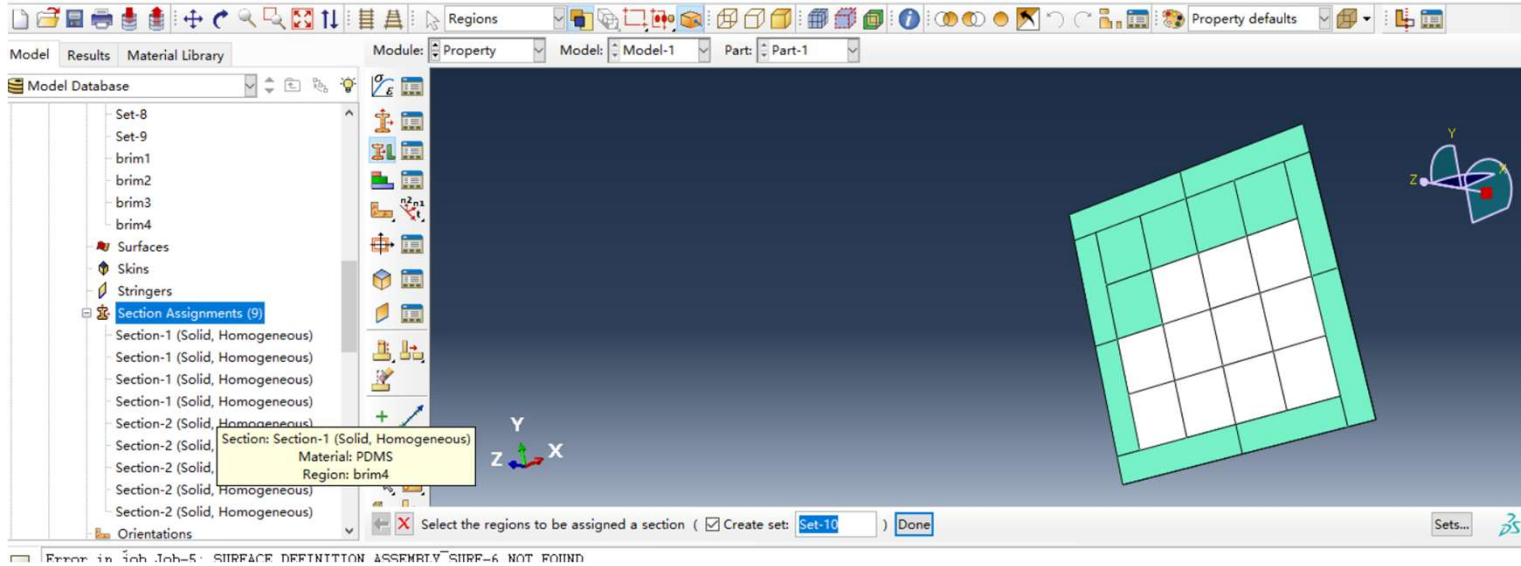
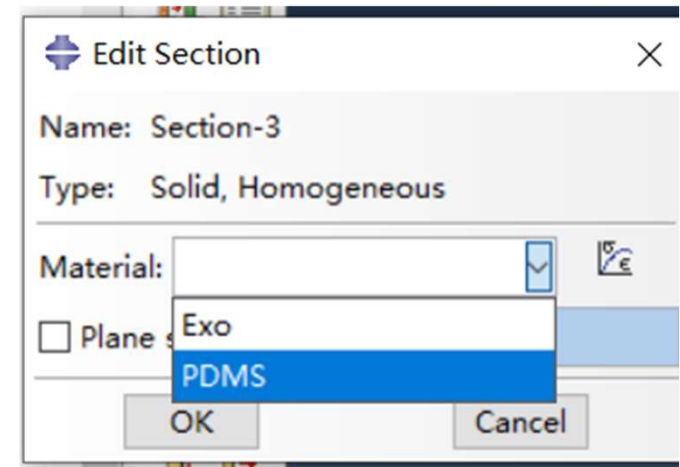
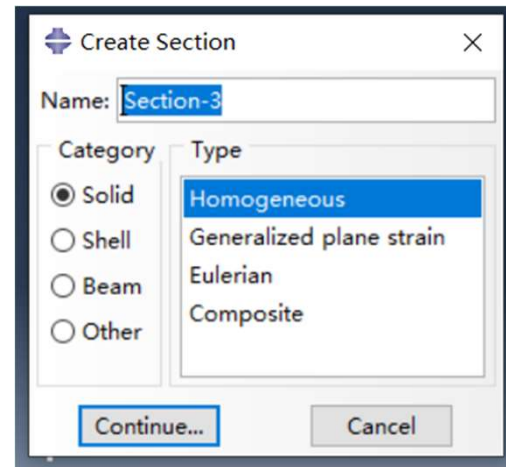
Figure 2





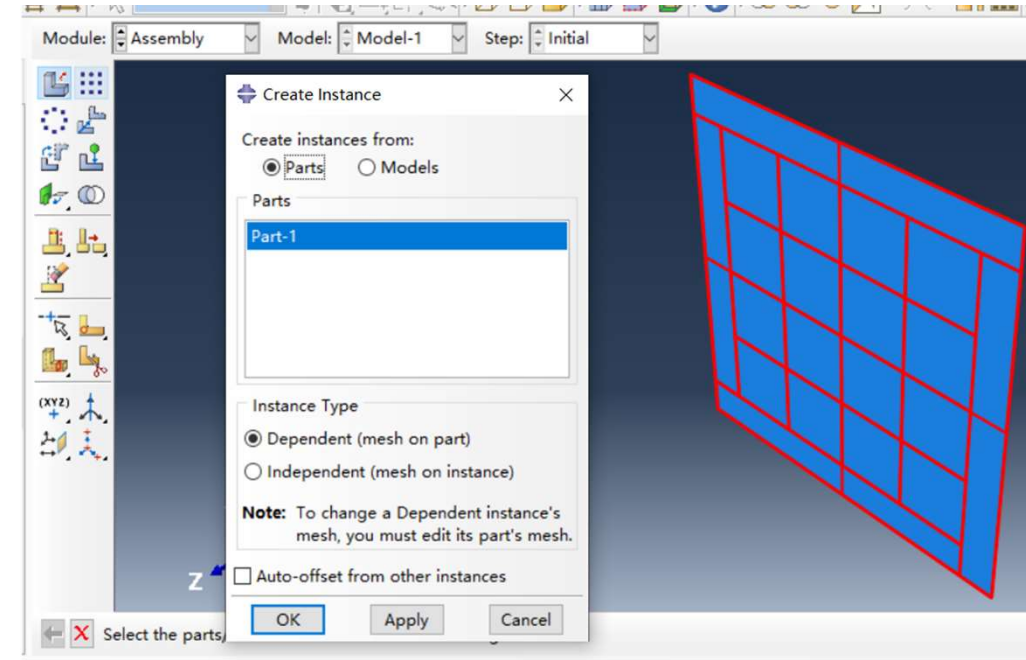
# Assign Section

- In Property Module, click Assign Section
- Select all your parts, then click mouse wheel to confirm
- In Edit Section Assignment window, select the section you just created, click “OK”
- After proper setting, all your model will be green

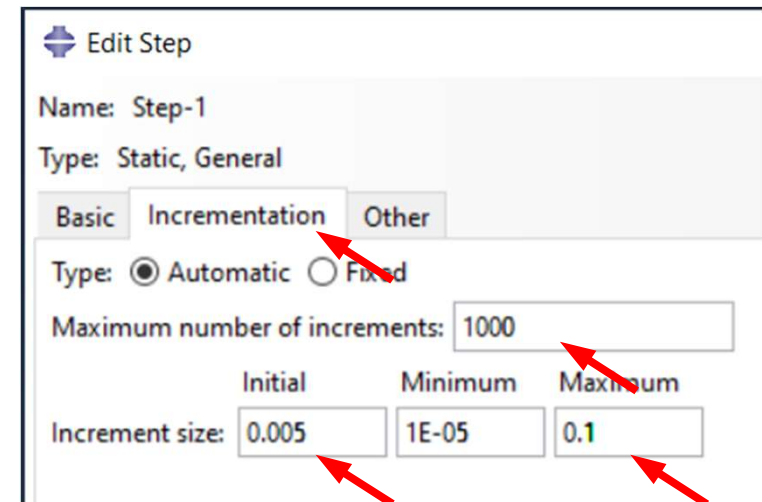
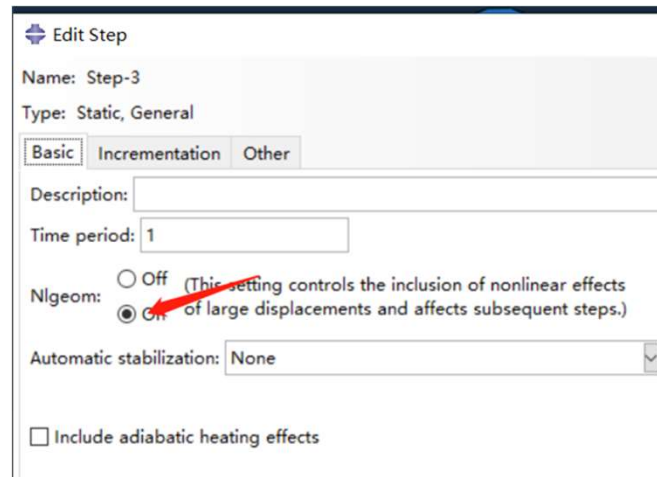
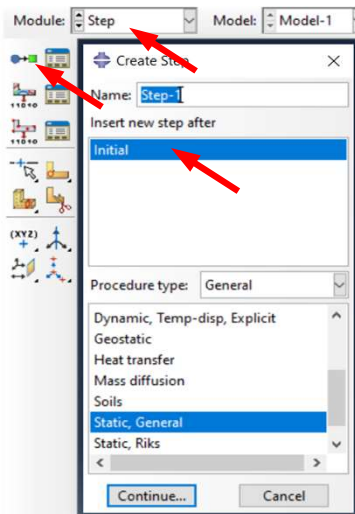


# Assembly => Step

- Click Module > Assembly > Create Instance
- After assembly, the part will become blue
- Click Module > Step > Create Step
- In Create Step window, select "Static, General", Click Continue
- In Edit Step window, click Basic > Nlgeom On
- Click Incrementation, change maximum number of increments = 1000, initial increment size = 0.005 and maximum increment size = 0.1
- Use the same method to create Step-2

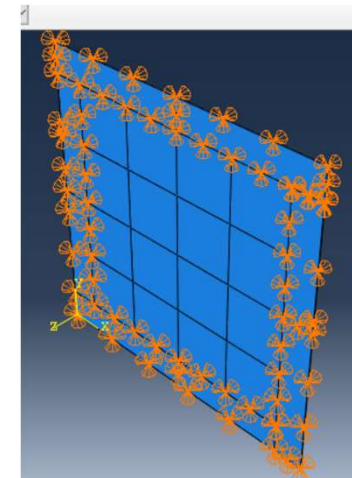
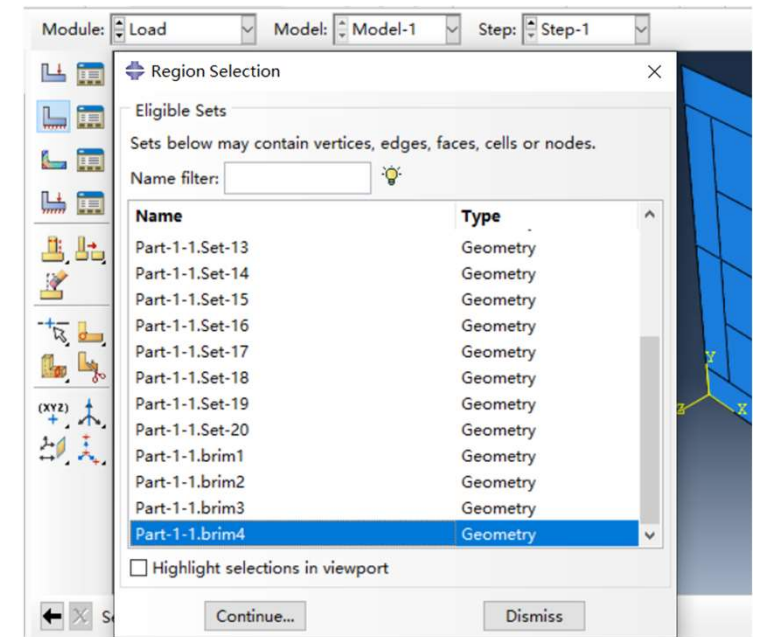
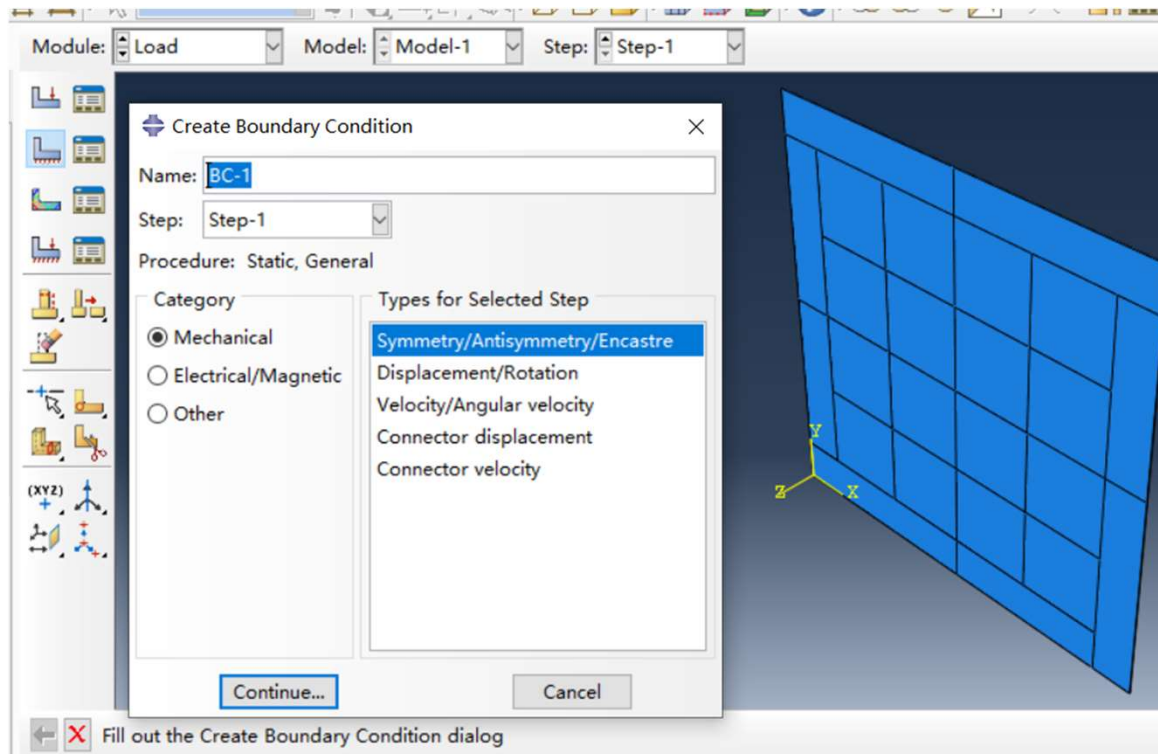


error aborted due to errors.  
1. Processes exited with an error.



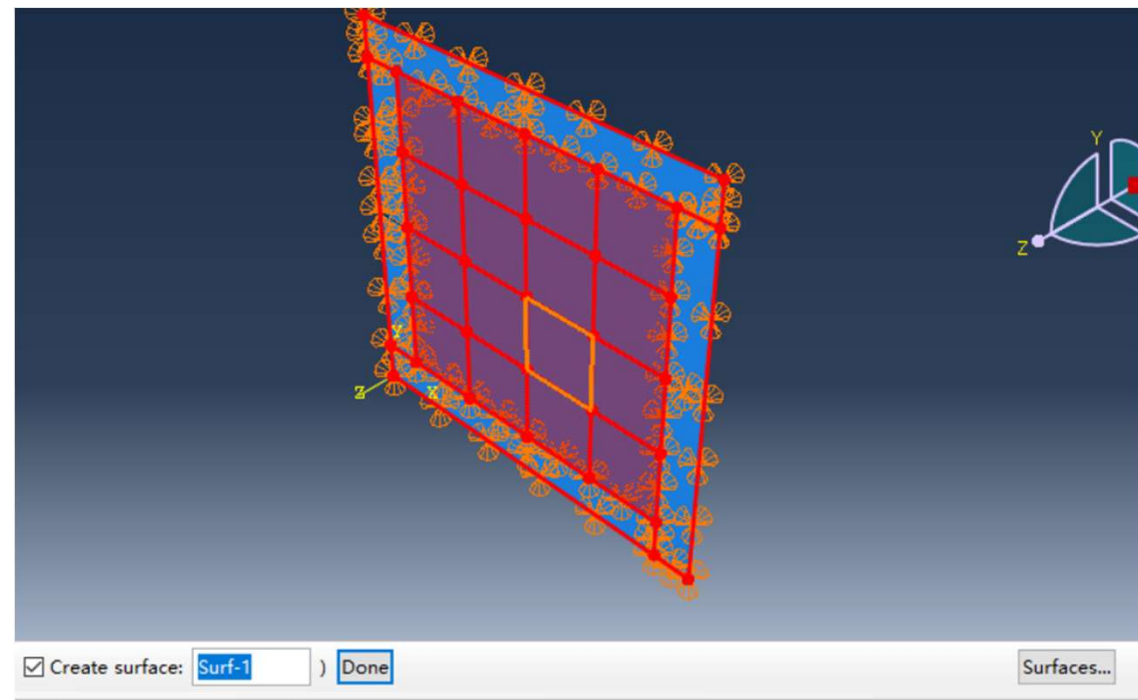
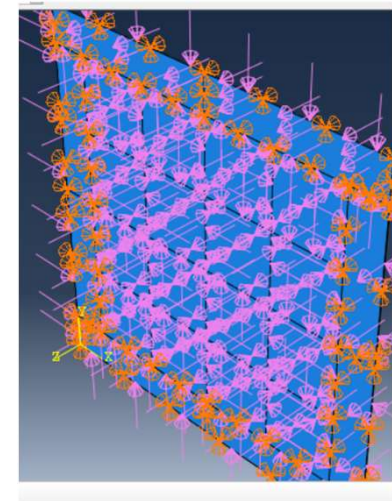
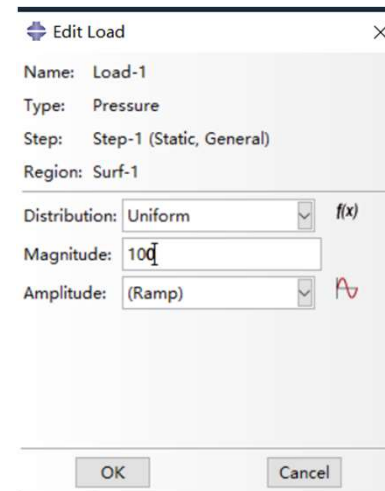
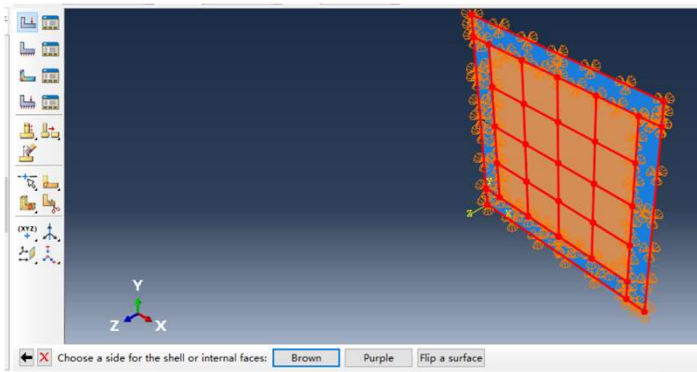
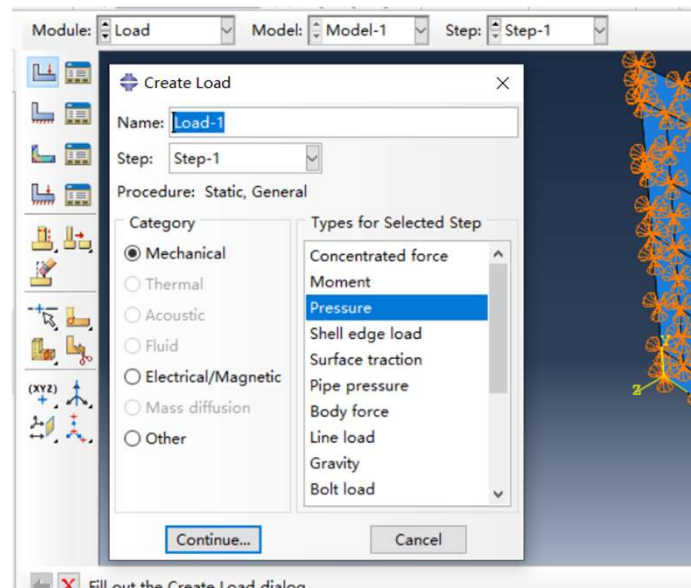
# Create Boundary Condition

- Click Module > Load > Create Boundary Condition
- In the Create Boundary Condition window, click Step > Step-1 > Mechanical >



# Create Load

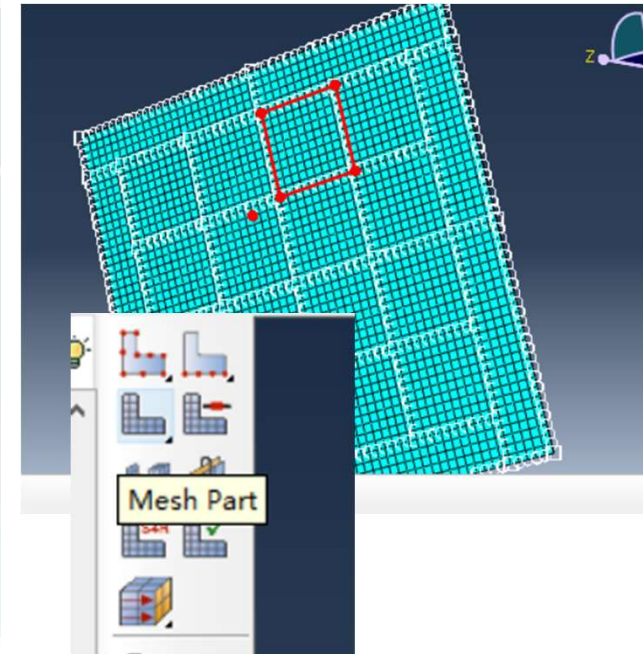
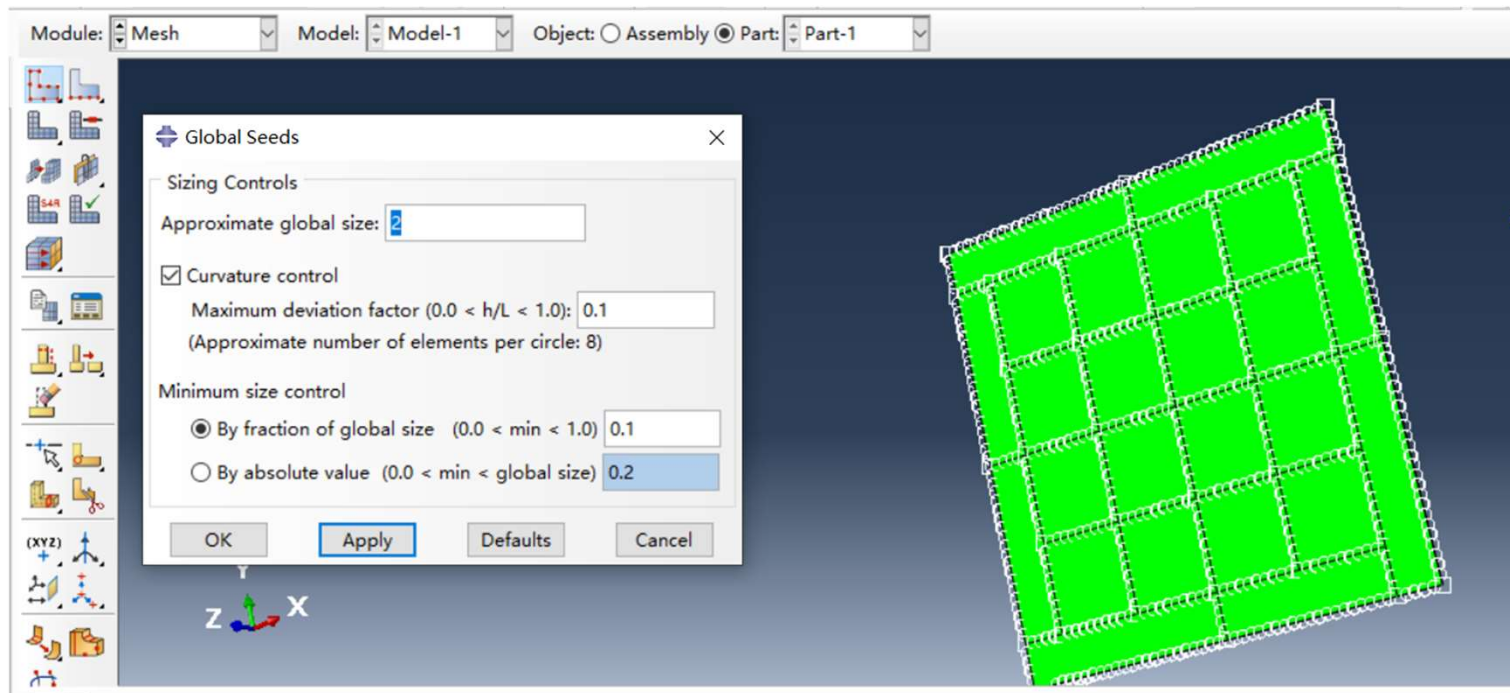
- In Load Module, click Create Load
- In Create Load window, choose the step of Step-1, click Mechanical > Pressure > Continue
- In Edit Load window, input Magnitude = 500,





# Mesh

- Click Module > Mesh > Object: Part
- After the part became pink, click Seed Part
- In Global Seeds window, set Approximate global size = 2, click "OK"



```
The model database has been saved to "C:\Users\jiao
Global seeds have been assigned.
2500 elements have been generated on part: Part-1
2500 elements have been generated on part: Part-1
```

# Job

- Click Module > Job > Create job (Fig.1)
- In Create Job window (Fig.1), create a name of this job, click “Continue”
- In Edit Job window (Fig.2), Click “OK”
- Click Job Manager (Fig.3), select the job and click “Submit”
- The “Status” will show “Submitted” followed by “Running”

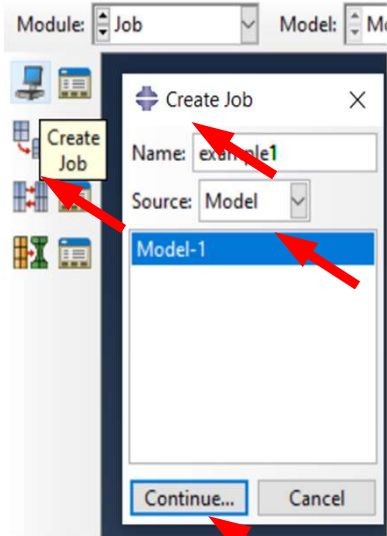


Figure 1

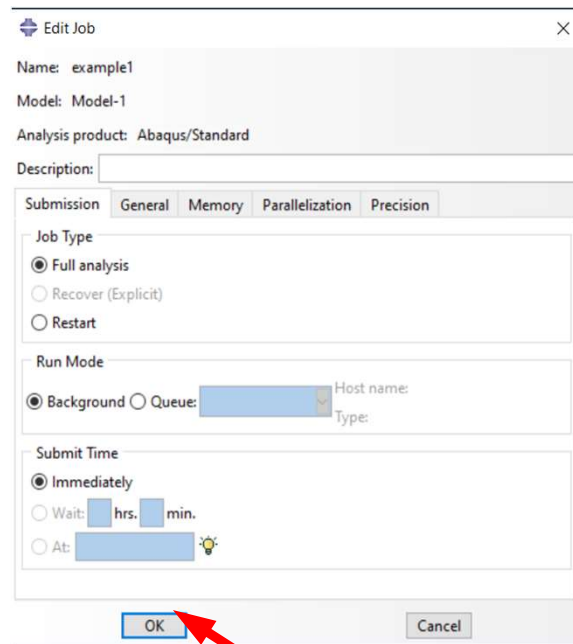


Figure 2

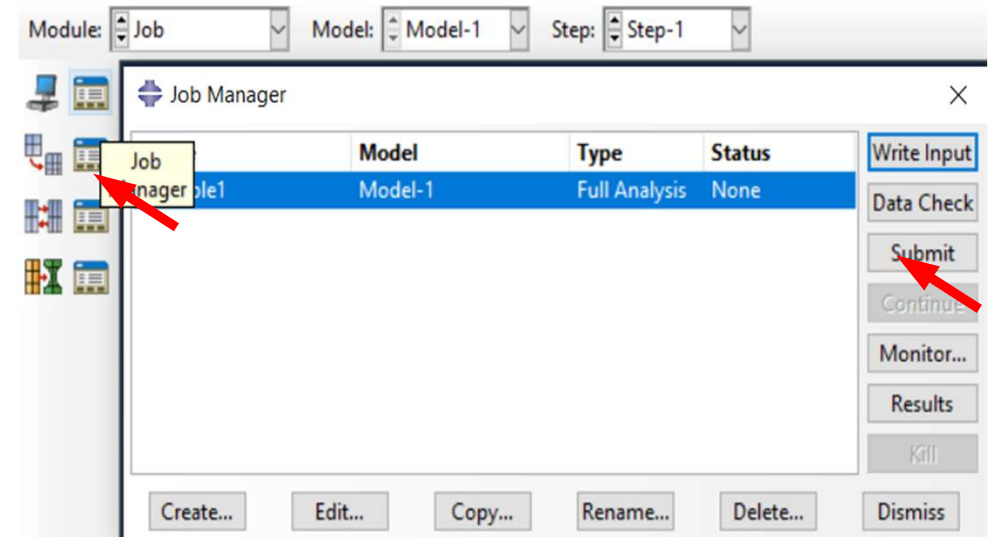


Figure 3



# Visualization

- After the job is completed, in job Manager window (Fig.1), click “Result”
- In Visualization Module (Fig.2), click “Plot Contours on Deformed Shape” to show the deformed shape with color gradient
- Click “Animate:Time History (Fig.2)” to show the animation
- Click “Animation Options (Fig.2)” to adjust animation speed

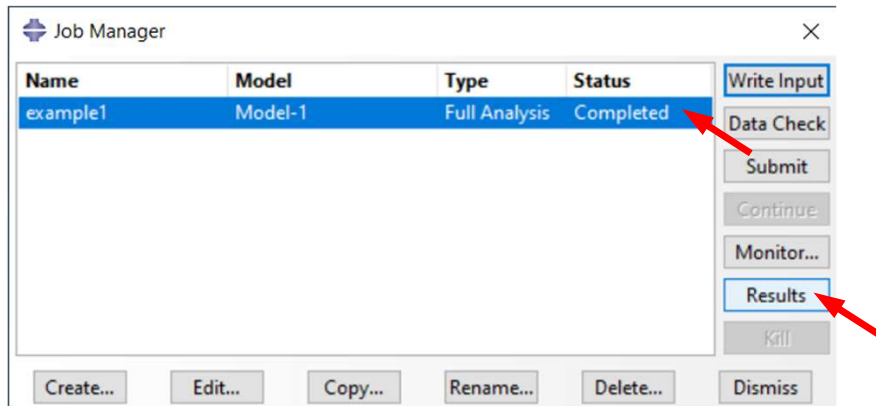


Figure 1

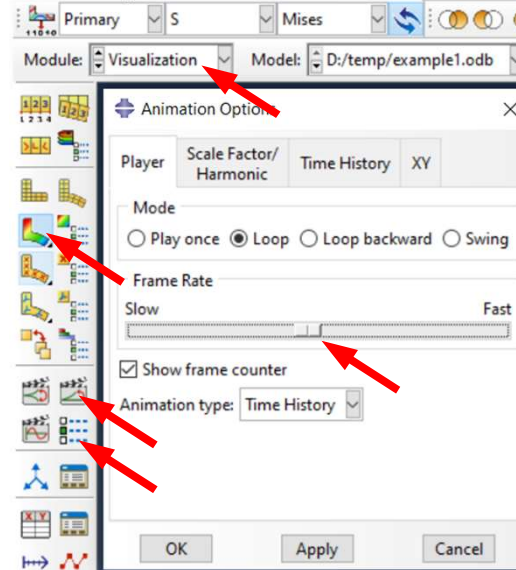
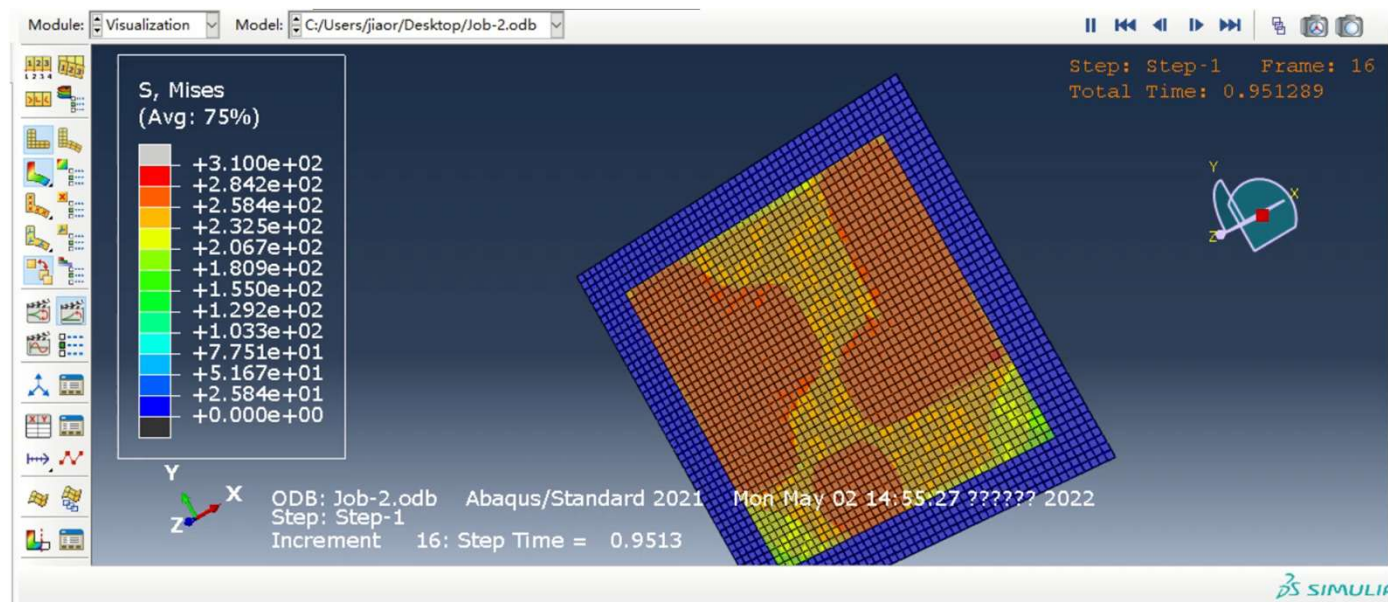
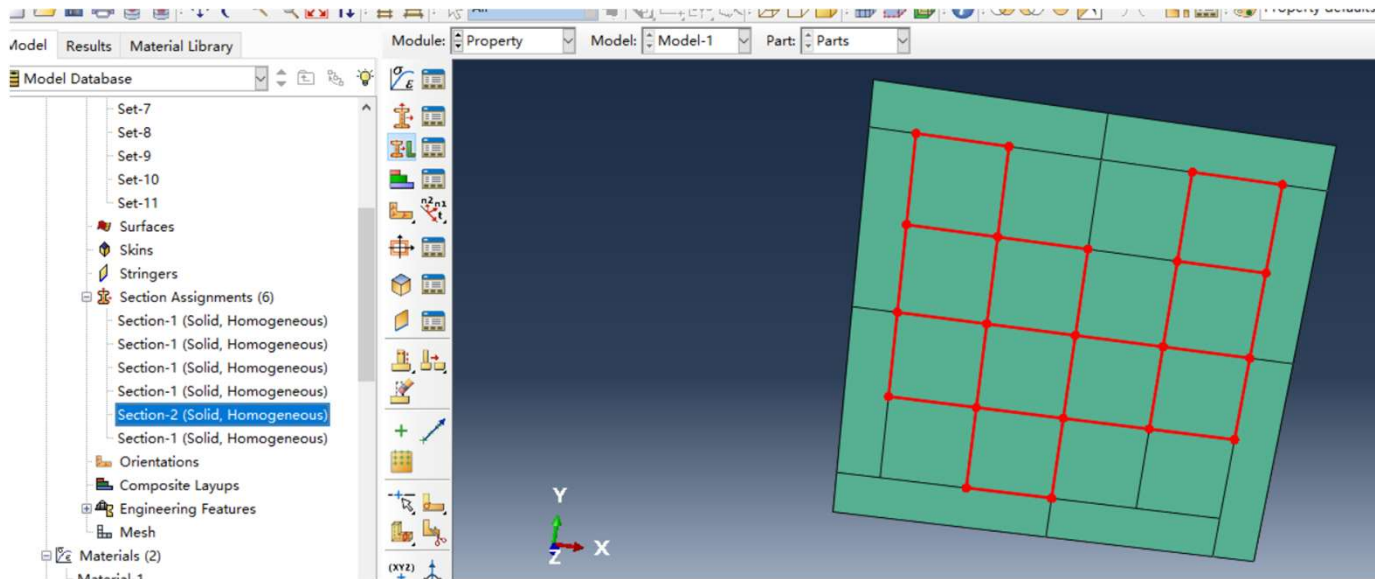


Figure 2

# Results: 1



## Results: 2

