

04

Static Structural Analysis

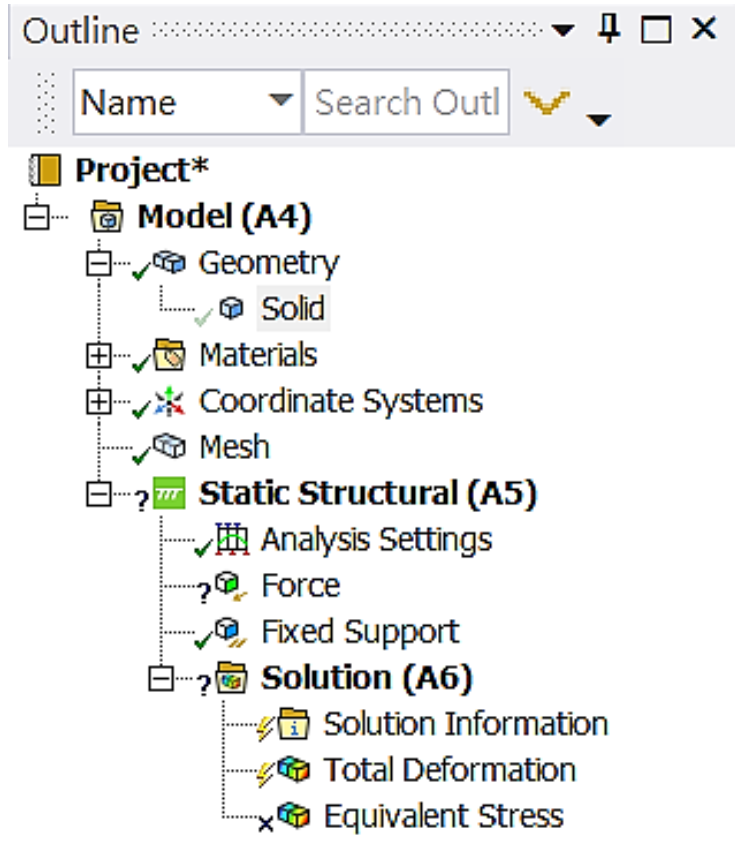
結構分析介紹



Introduction of ANSYS Workbench

■ 結構樹狀態顯示

	A
1	Static Structural
2	Engineering Data
3	DM Geometry
4	Model
5	Setup
6	Solution
7	Results



說明分支全部被定義



說明輸入的數據不完整



說明需要求解



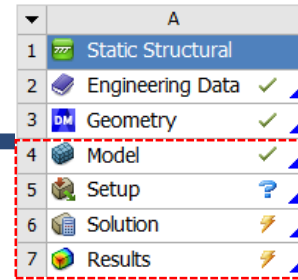
說明被抑制，不能被求解



說明體積或零件被隱藏

Introduction of ANSYS Workbench

■ Detail視窗



Details of "Force" ▾ 🔍 □ ×

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
☒ X Component	0. N (ramped)
☒ Y Component	0. N (ramped)
☒ Z Component	0. N (ramped)
Suppressed	No

Details of "Total Deformation" ▾ 🔍 □ ×

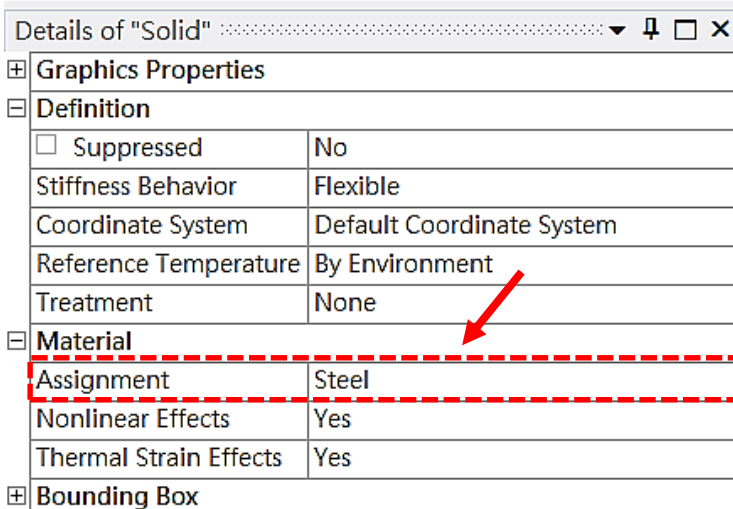
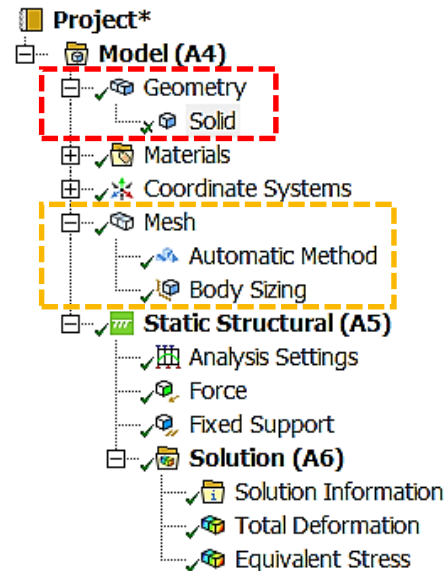
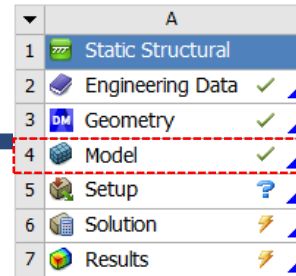
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Definition	
Type	Total Deformation
By	Time
☐ Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No
Results	
☐ Minimum	0. mm
☐ Maximum	2.6374e-003 mm

白色區域	可編輯的資料設定
灰色區域	不可編輯的資料，僅供顯示信息數據
黃色區域	未完成的資料設定(待輸入)
粉色區域	尚未更新的結果數據(需重新求解)

Introduction of ANSYS Workbench

■ Model

- **Geometry:** 模型材料給定
- **Mesh:** 網格分割



Introduction of ANSYS Workbench

■ 邊界條件給定

A	
1	Static Structural
2	Engineering Data
3	DM Geometry
4	Model
5	Setup
6	Solution
7	Results

The screenshot displays the ANSYS Workbench interface. The 'Setup' tab is selected in the top ribbon, and the 'Insert' menu is open, showing various boundary conditions and loads. The 'Outline' pane on the left shows the project hierarchy, with 'Static Structural (A5)' selected. The 'Details of "Static Structural (A5)"' pane at the bottom shows the 'Definition' section with 'Physics Type' set to 'Structural Mechanics' and 'Solver Target' set to 'Mechanical APDL'.

Insert Menu Options:

- Acceleration
- Standard Earth Gravity
- Rotational Velocity
- Rotational Acceleration
- Pressure
- Hydrostatic Pressure
- Force
- Remote Force
- Bearing Load
- Bolt Pretension
- Moment
- Line Pressure
- Thermal Condition
- Joint Load
- Fixed Support
- Displacement
- Remote Displacement
- Frictionless Support
- Compression Only Support
- Cylindrical Support

Details of "Static Structural (A5)"

Definition	
Physics Type	Structural Mechanics
Analysis Type	Static
Solver Target	Mechanical APDL

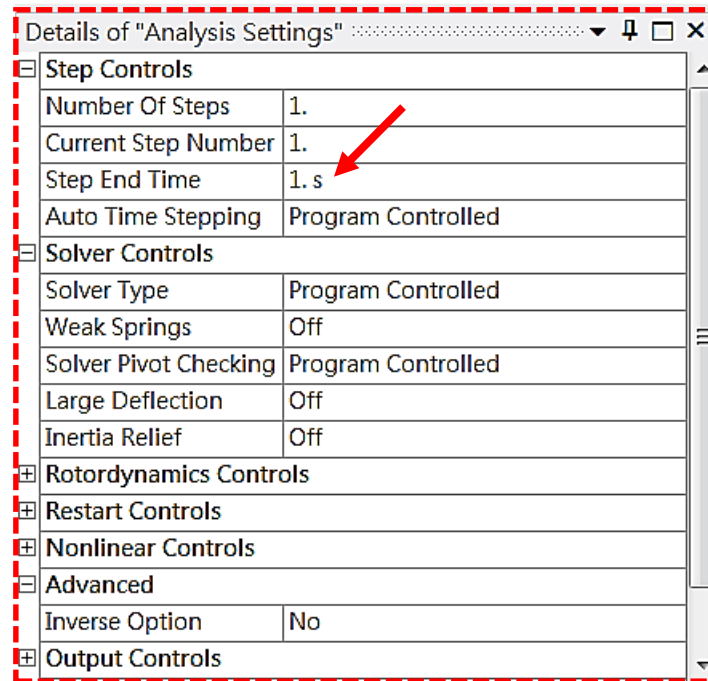
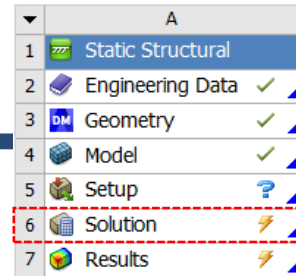
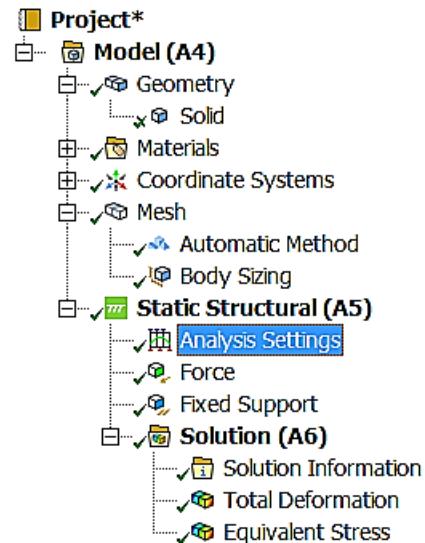
Introduction of ANSYS Workbench

■ 解題條件設定

- Step Control
- Number of steps
- Current Step Number
- Step Ends

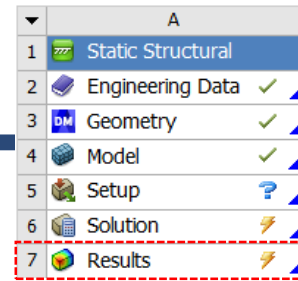
■ Solver Control

- 解題形式
- 大變形等



Introduction of ANSYS Workbench

■ 後處理(Post-processing)



設定顯示方式

選取結果項目

Contour設定

- 自訂數值
- 增加色塊
- 自訂色彩

ANSYS 2019 R3

0.00 35.00 70.00 (mm)

17.50 52.50

Graph

Animation

20 Frames 2 Sec (Aut)

Tabular Data

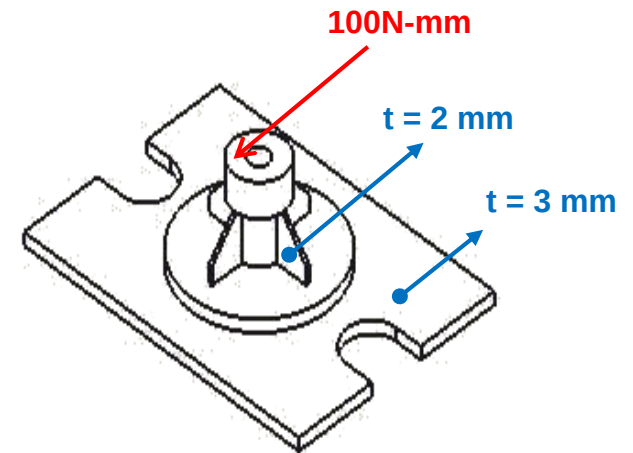
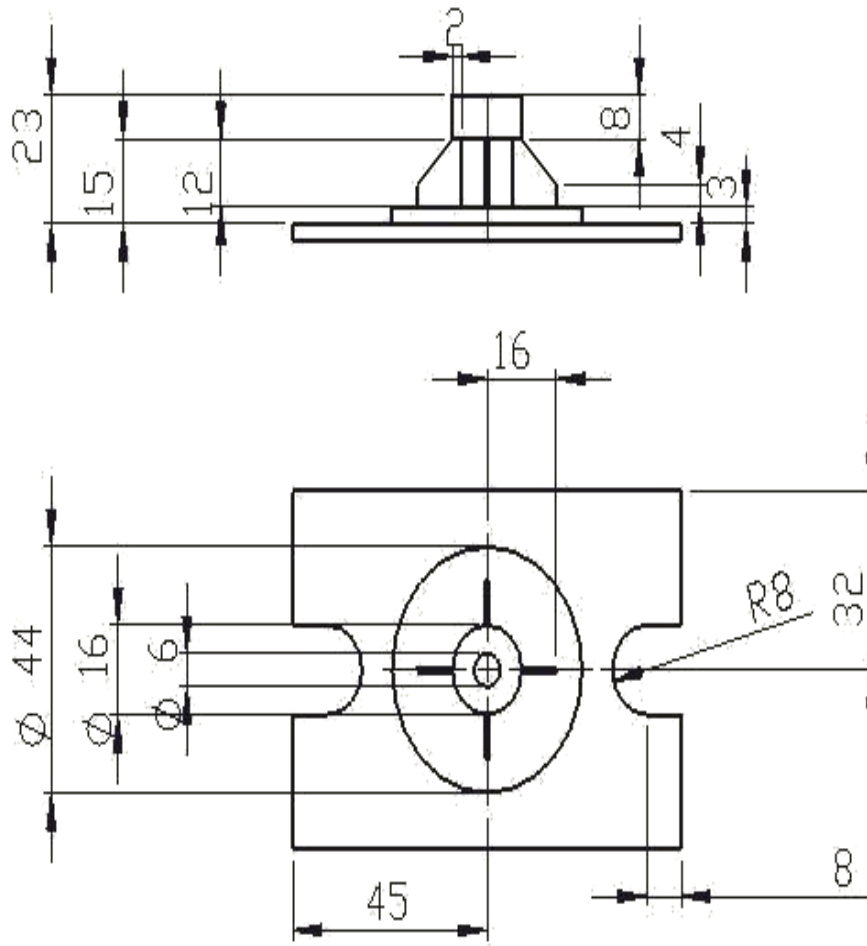
Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1.	0.	2.6374e-003	3.672e-004

Ready

No Messages No Selection Metric (mm, kg, N, s, mV, mA) Degrees rad/s Celsius

3D Solution – Ex.5 (來源：ANSYS Workbench有限元分析從入門到精通)

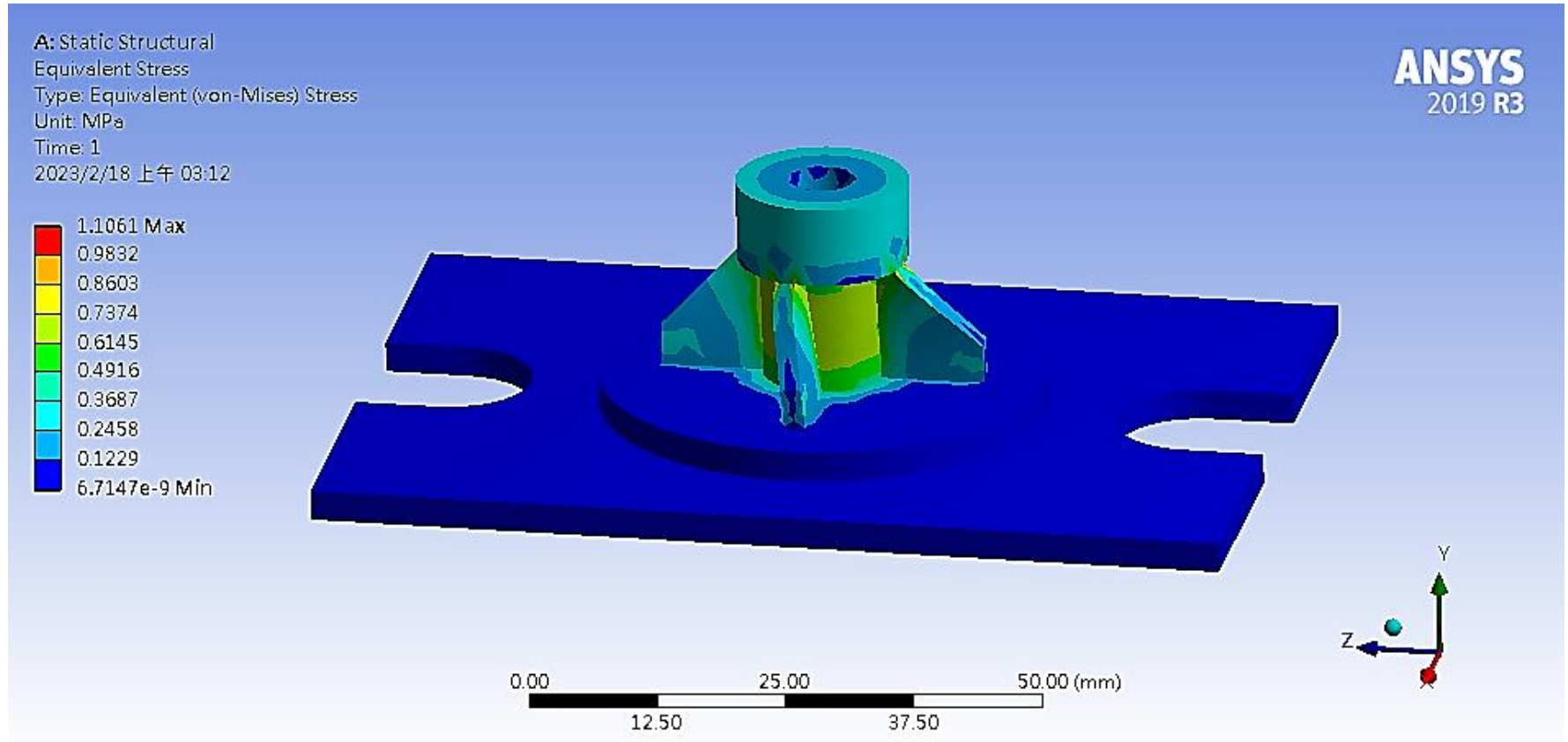
試建構機蓋模型，尺寸如圖所示，於中央孔頂端面給一100N-mm順時針方向扭轉，觀察其等效應力，材料選用鋼，設定ELEMENT SIZE為3的MESH。



3D Solution – Ex.5

- 學習目標
- Revolve
 - Pattern
 - Mesh-Sizing
 - 解題步驟
 - 基本後處理

試建構機蓋模型，尺寸如圖所示，於中央孔頂端面給一100N-mm順時針方向扭轉，觀察其等效應力，材料選用鋼，設定ELEMENT SIZE為3的MESH。



等效應力
Equivalent Stress

Mesh



■ Mesh Approach

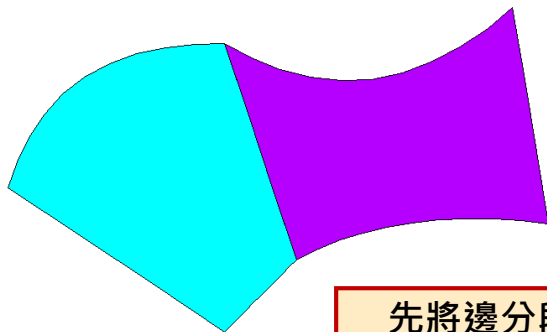
➤ Solid modeling

- ✓ Free mesh
- ✓ Mapped mesh
- ✓ ...

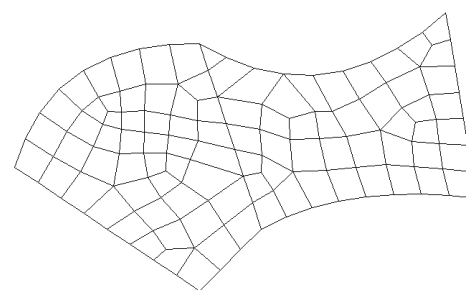
網格控制方法非常多元，實際情況依模型、受力狀態、邊界條件而定

	節點數：4 節點自由度： UX, UY (u, v)
	節點數：8 節點自由度： UX, UY (u, v)

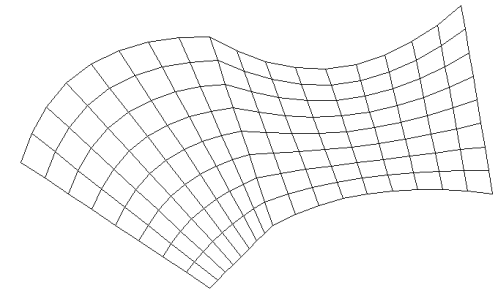
Solid Model



先將邊分段



Free Mesh

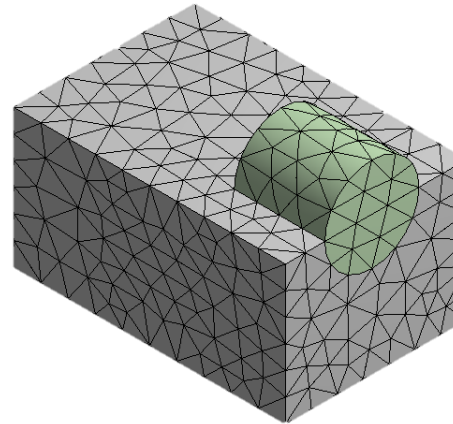
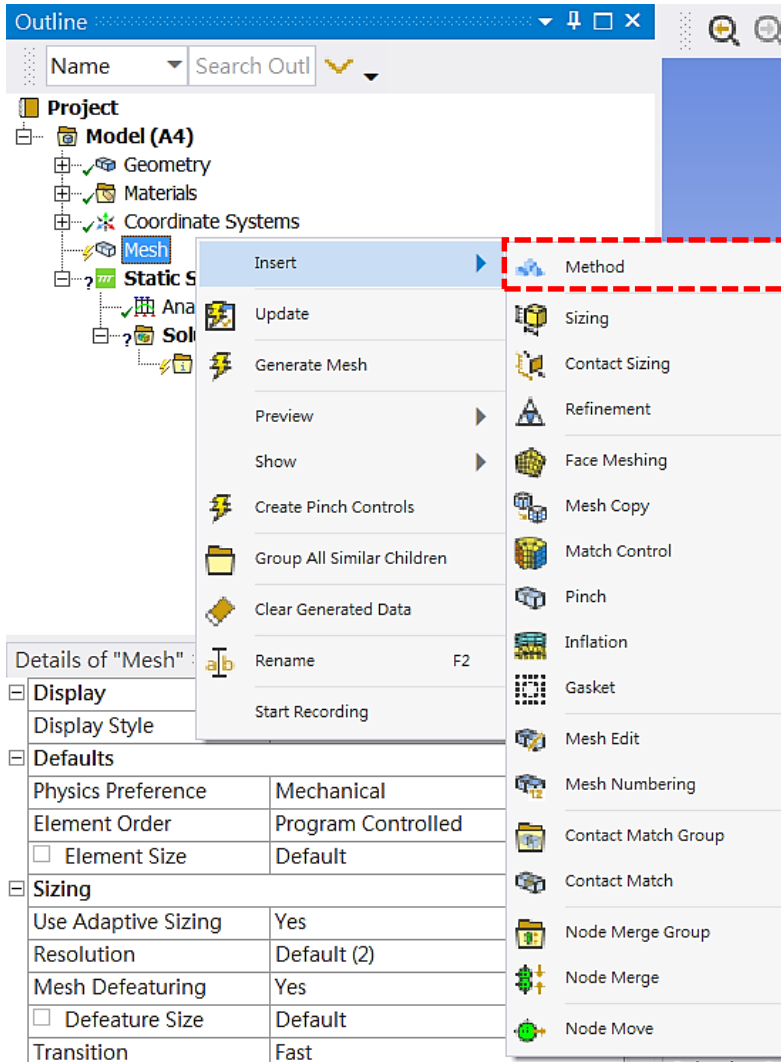
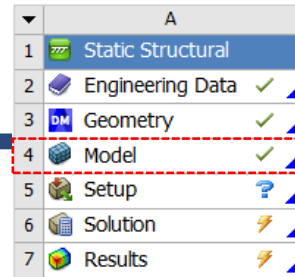


Mapped Mesh

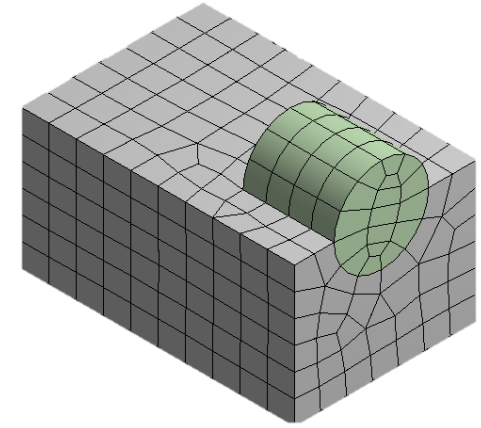
Introduction of ANSYS Workbench

■ 局部網格處理

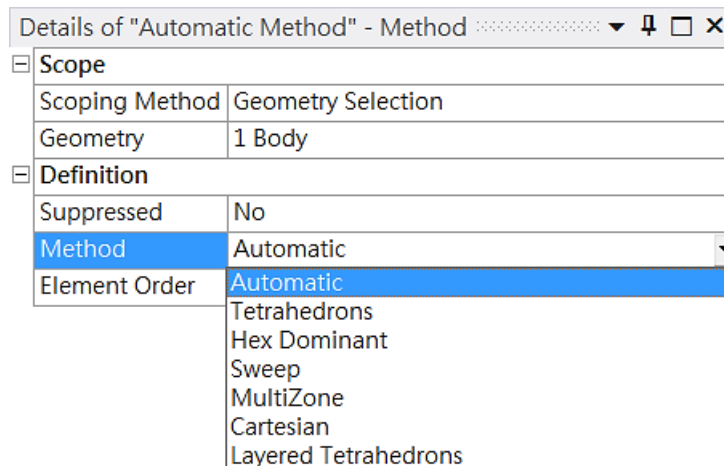
➤ Method



四面體



六面體



Introduction of ANSYS Workbench

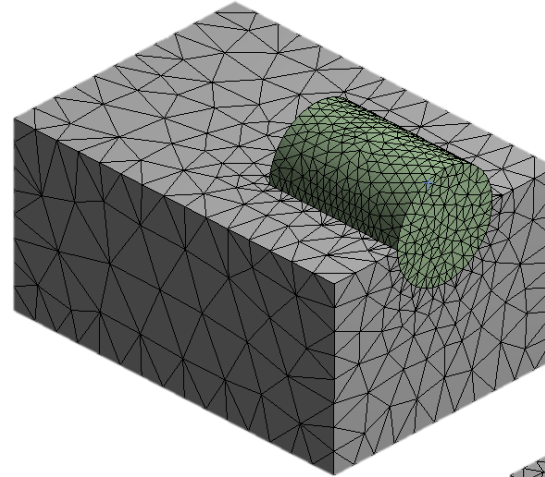
■ 全域網格處理

➤ Sizing

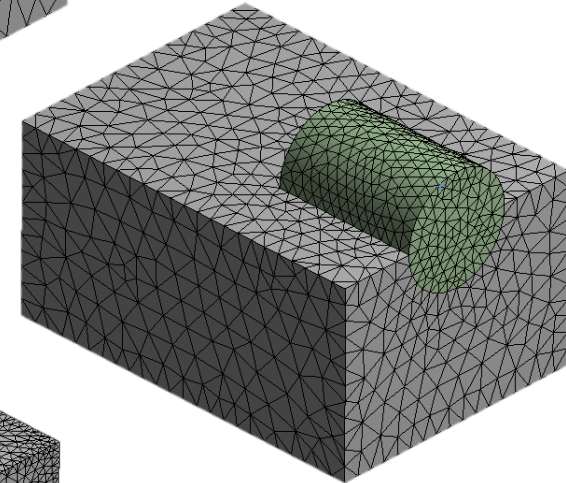
✓ Resolution

Details of "Mesh"	
Display	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
☐ Element Size	Default
Sizing	
Use Adaptive Sizing	Yes
Resolution	Default (2)
Mesh Defeaturing	Yes
☐ Defeature Size	Default
Transition	Fast
Span Angle Center	Coarse
Initial Size Seed	Assembly
Bounding Box Di...	5.2715 in
Average Surface ...	1.4345 in ²
Minimum Edge L...	0.61842 in
Quality	
Inflation	
Advanced	
Statistics	

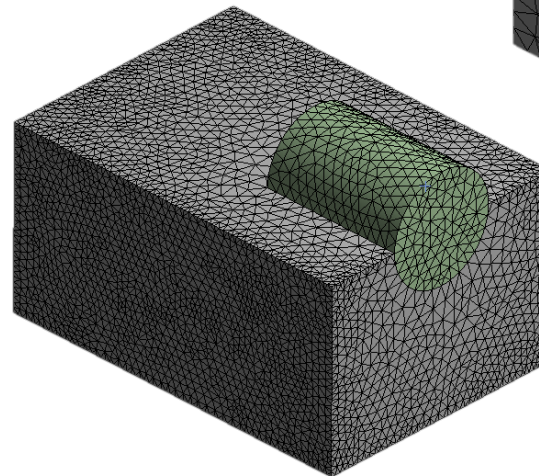
A	
1	Static Structural
2	Engineering Data
3	Geometry
4	Model
5	Setup
6	Solution
7	Results



Resolution = 2



Resolution = 4

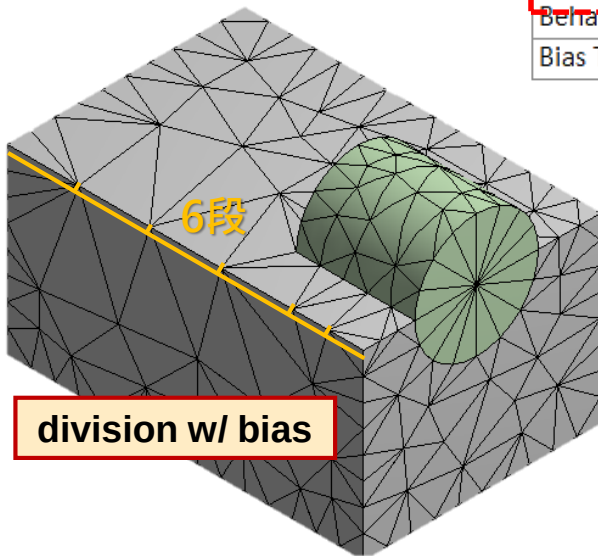
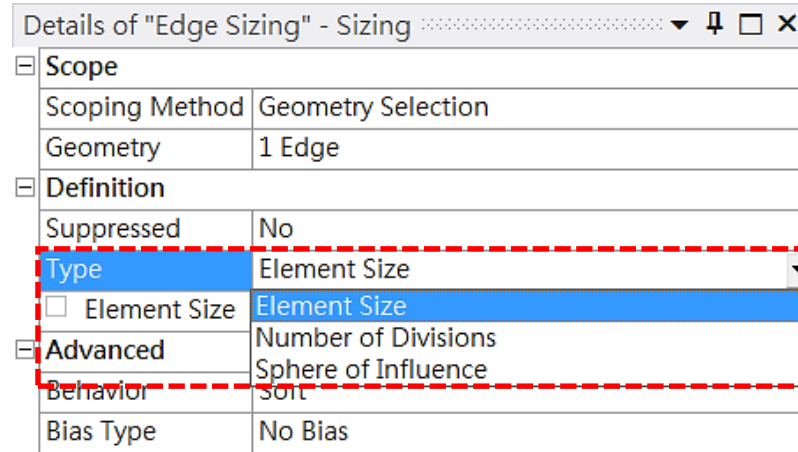
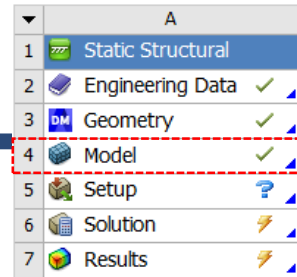


Resolution = 7

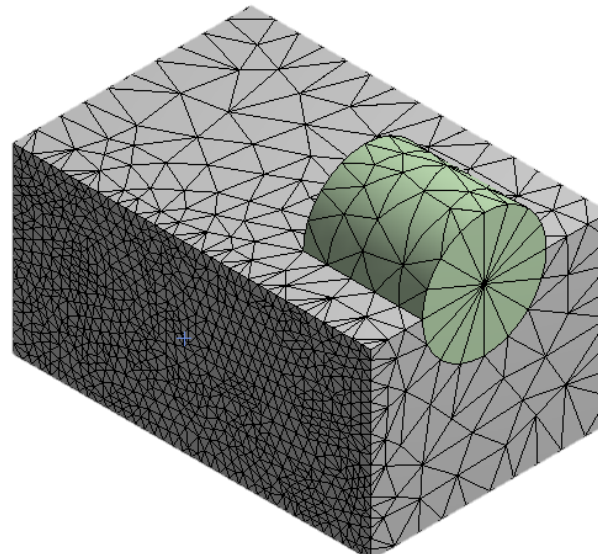
Introduction of ANSYS Workbench

■ 局部網格處理

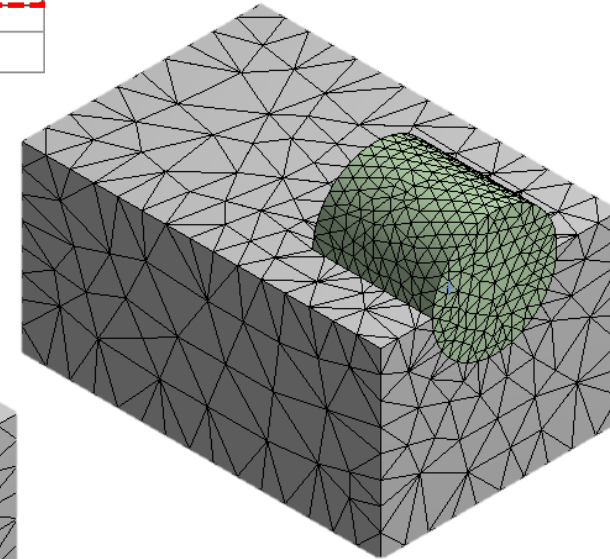
➤ Sizing



Edge



Face



Body

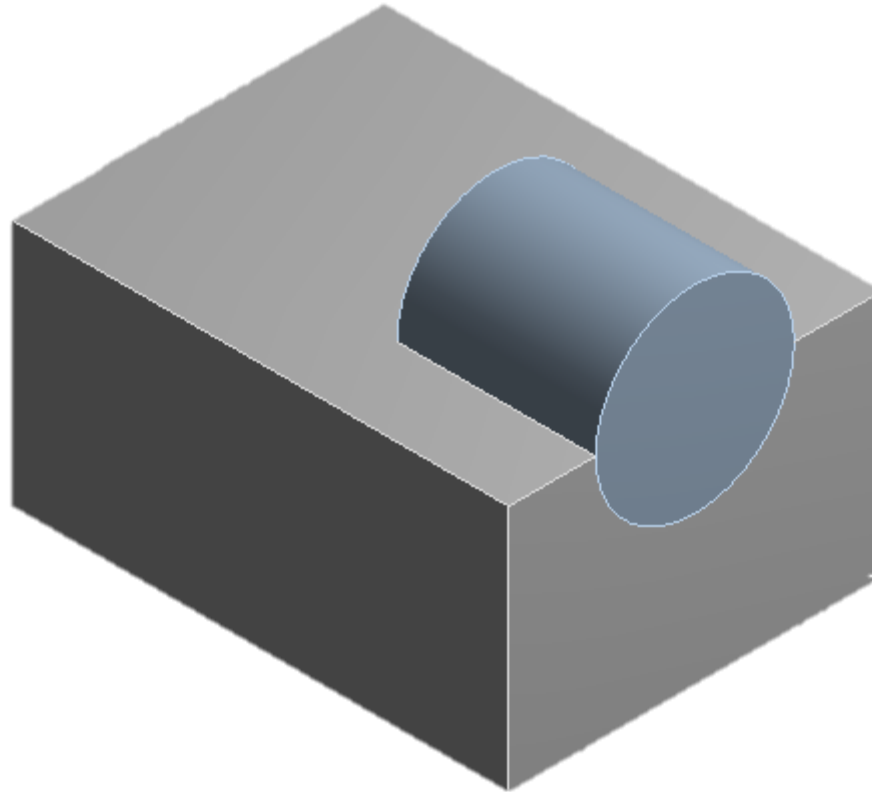


Mesh – Ex.6

試繪製模型如圖所示，方塊長寬高分別為20mm、15mm、10mm，圓柱半徑4mm、深8mm，請以不同功能進行網格化練習(Method、Resolution、指定不同幾何之Sizing)

學習目標

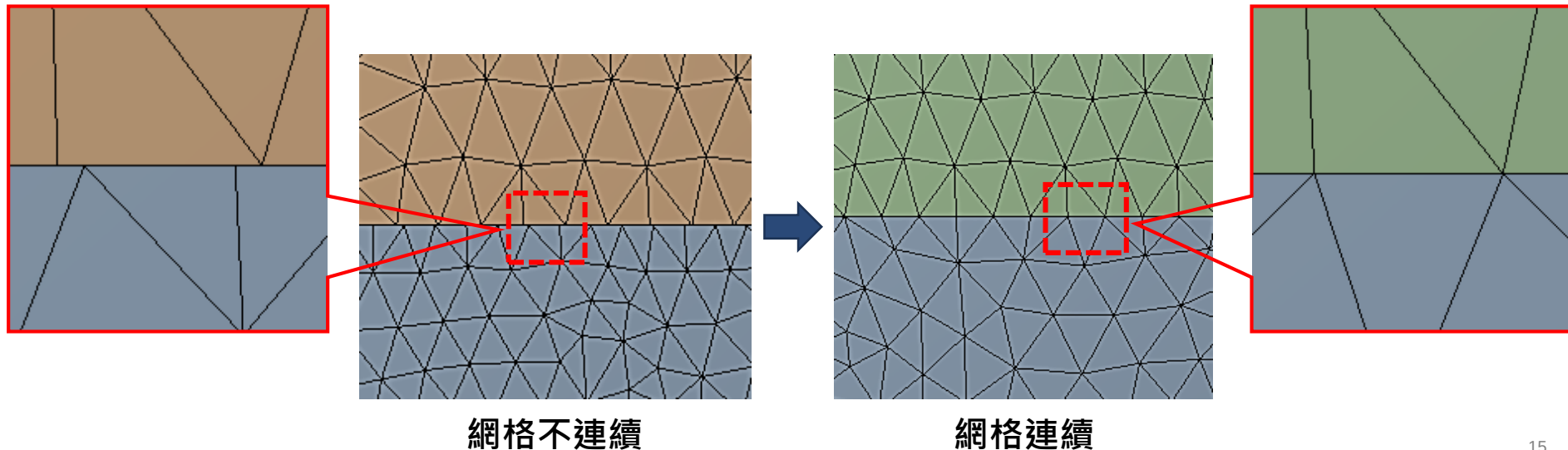
- Add frozen
- Boolean
- Mesh-Method
- Mesh-Resolution
- Mesh-局部Sizing





Mesh

- **元素的連續性**，相鄰元素需使用共同的節點和自由度
- 三角形和契形元素可被使用於過度區
- 元素需盡可能保持其原來的形狀，即不能扭曲太嚴重
- 在**施力處**的網格分割需良好
- 在**預期應力集中處**如孔洞、凹槽等處的分割，元素尺寸需較小且分佈良好
- 網格的分割的密度需盡可能隨應力分佈而調整

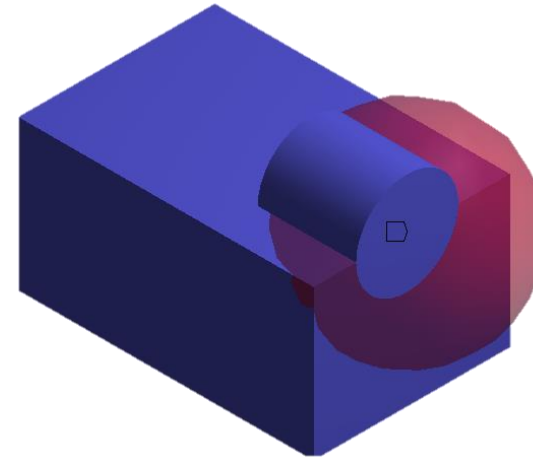
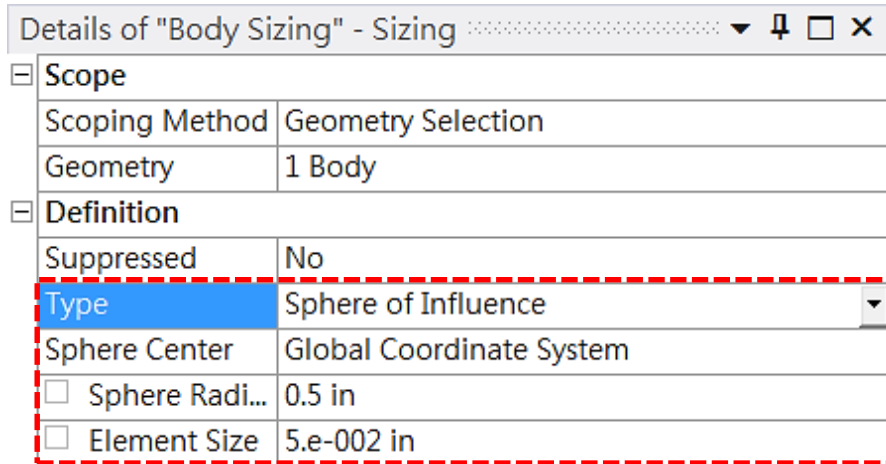
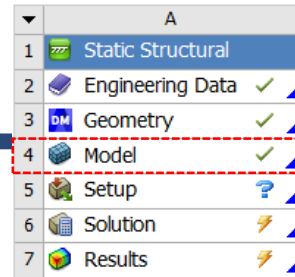


Introduction of ANSYS Workbench

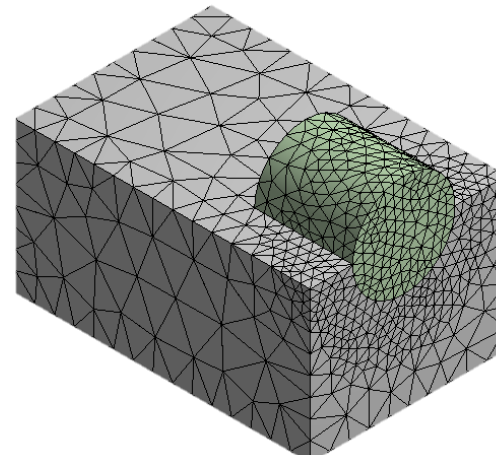
■ 局部網格處理

➤ Sizing

✓ Sphere of influence (Pinball)



Pinball

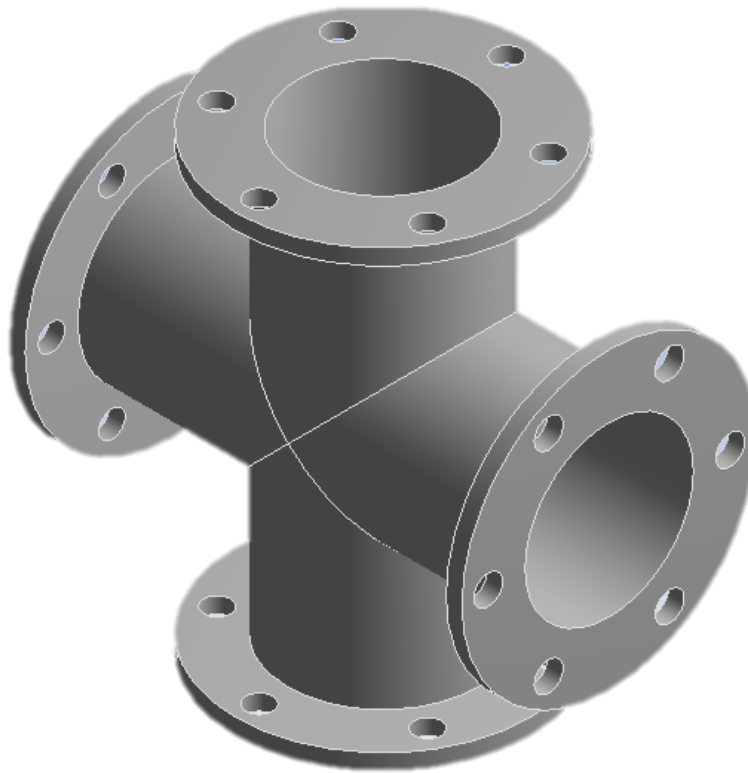


Pinball mesh

Mesh – Ex.7 (來源：ANSYS Workbench有限元分析從入門到精通)



請依下列實體模型(pipe.agdb)進行不同功能之網格化練習



Mesh – Ex.7

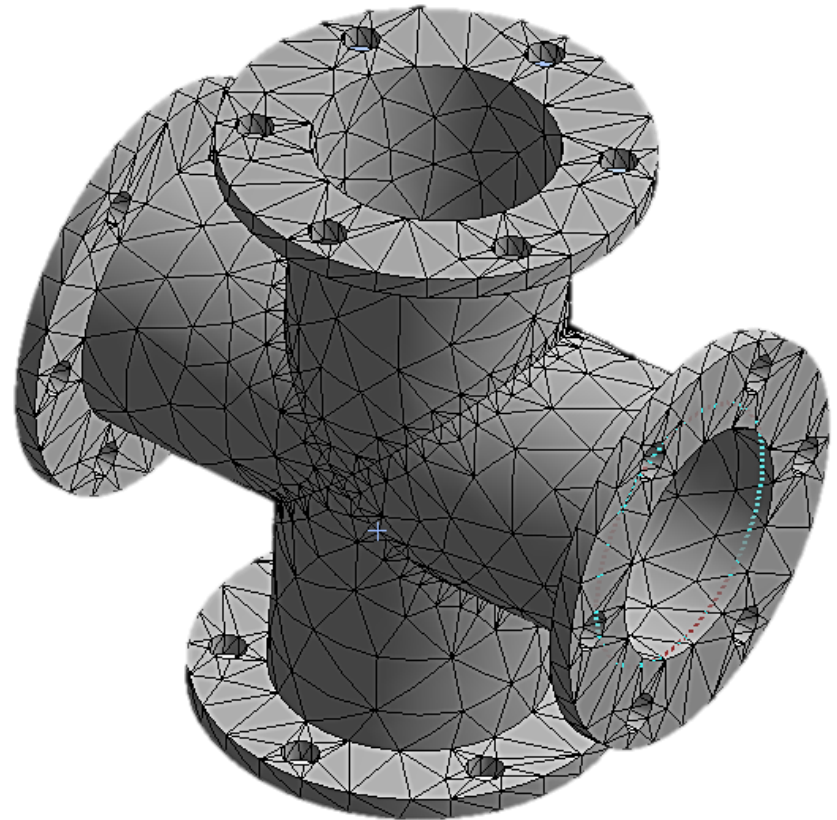
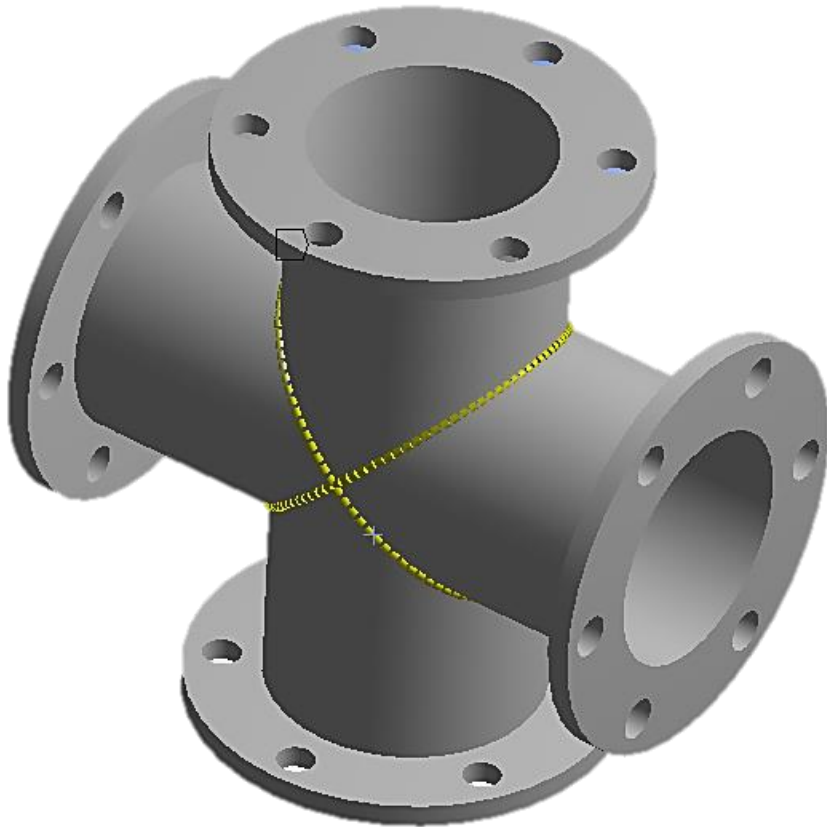
學習目標

- Mesh-Division
- Mesh-Refinement
- Mesh-Pinball
- 剖面觀察

■ 局部網格A

➤ Sizing

- ✓ Number of division



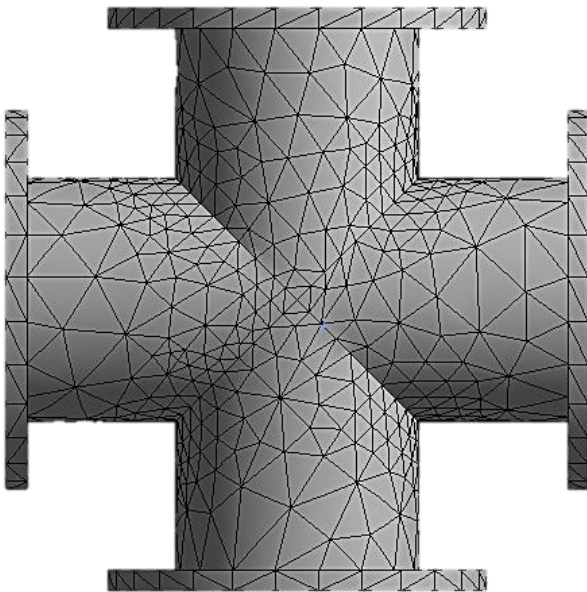
Mesh – Ex.7

學習目標

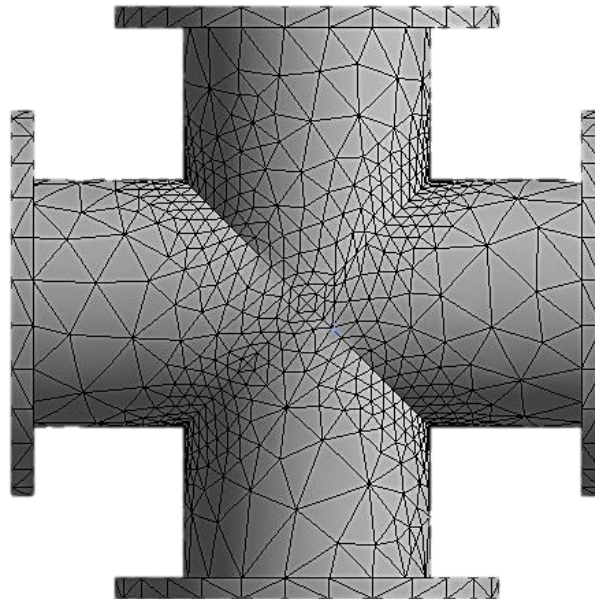
- Mesh-Division
- Mesh-Refinement
- Mesh-Pinball
- 剖面觀察

■ 局部網格B

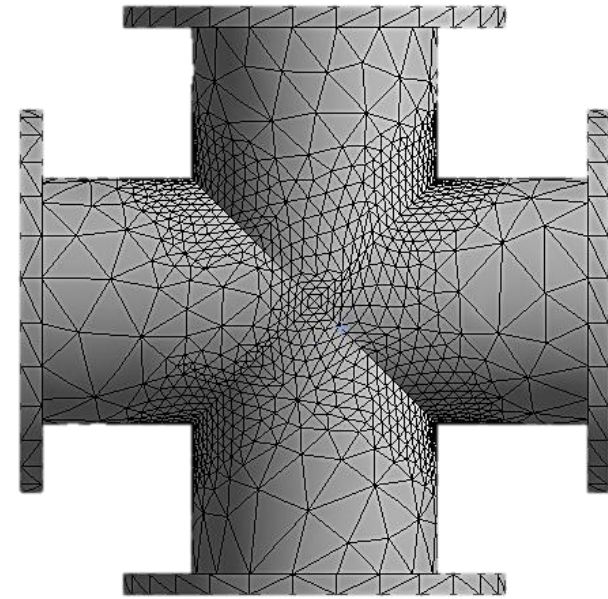
➤ Refinement



Refinement = 1



Refinement = 2



Refinement = 3

Mesh – Ex.7

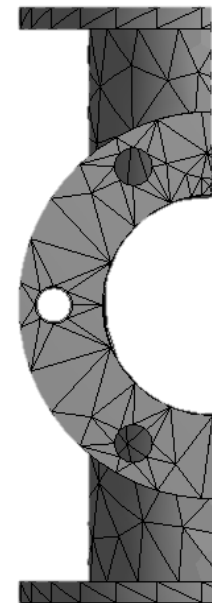
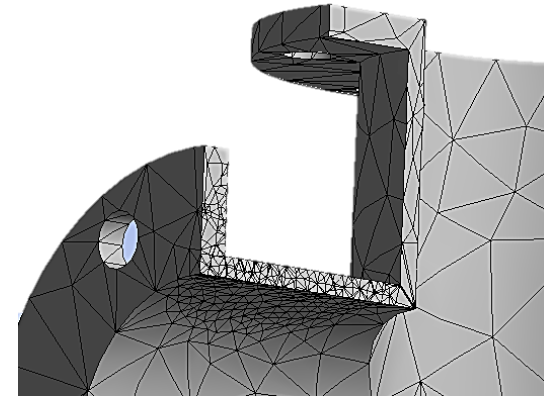
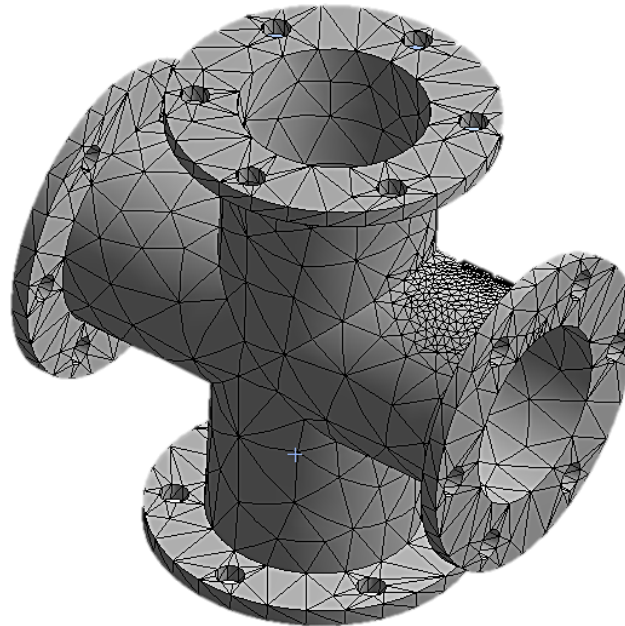
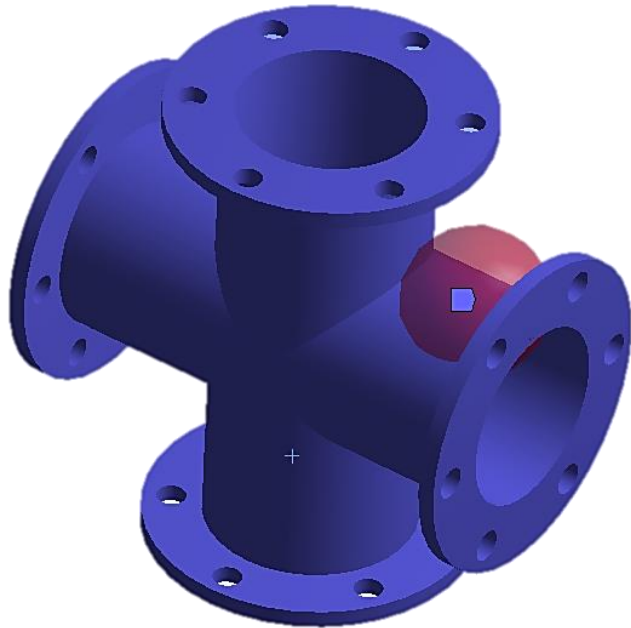
學習目標

- Mesh-Division
- Mesh-Refinement
- Mesh-Pinball
- 剖面觀察

■ 局部網格C

➤ Sizing

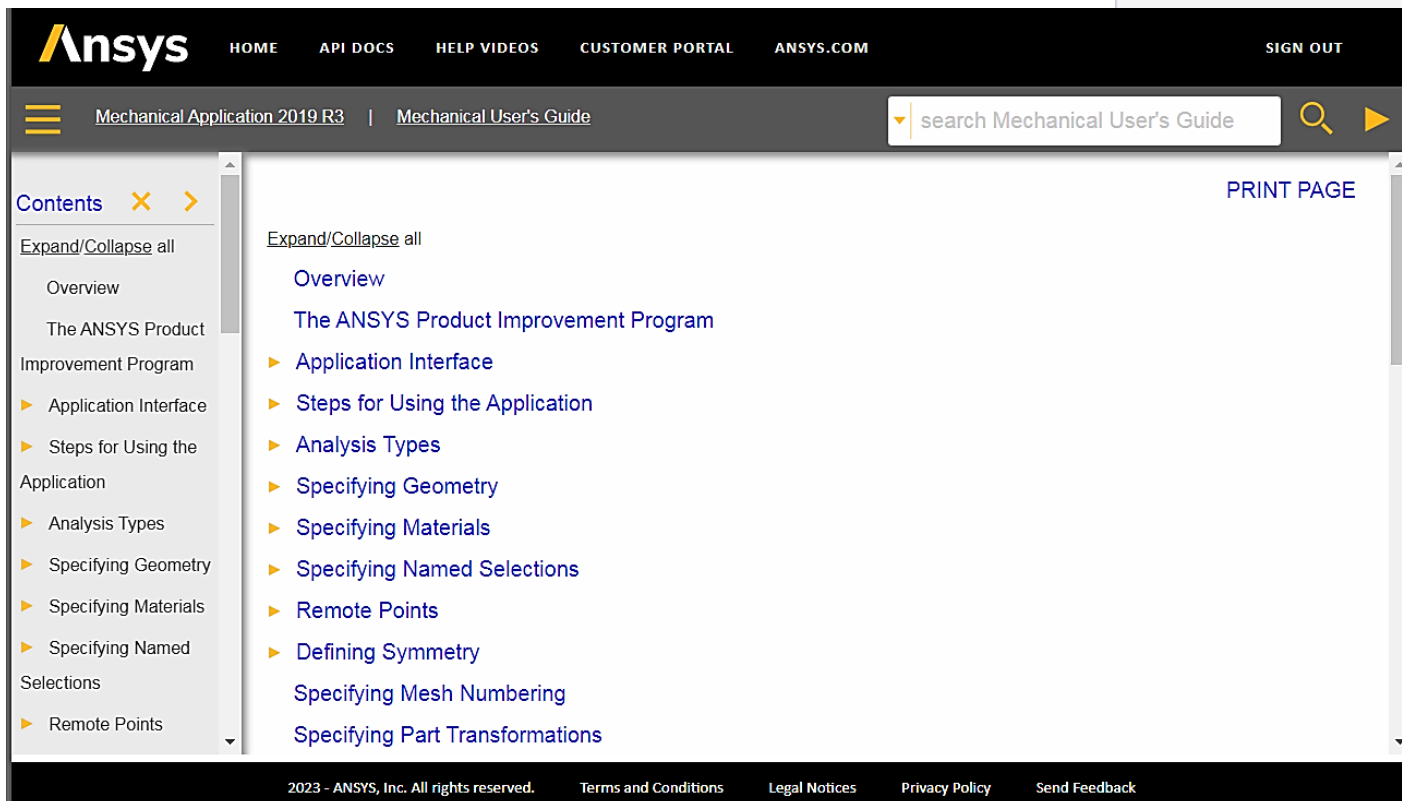
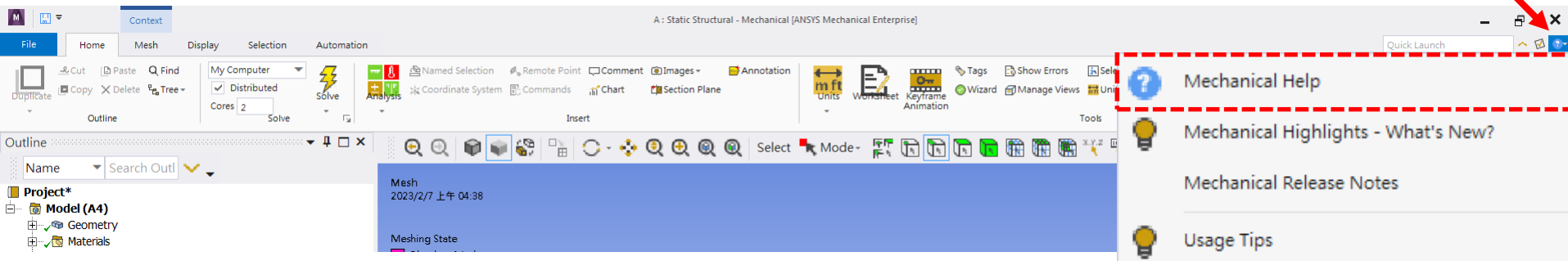
- ✓ Sphere of influence (Pinball)



Introduction of ANSYS Workbench



■ 複雜網格設定→查HELP

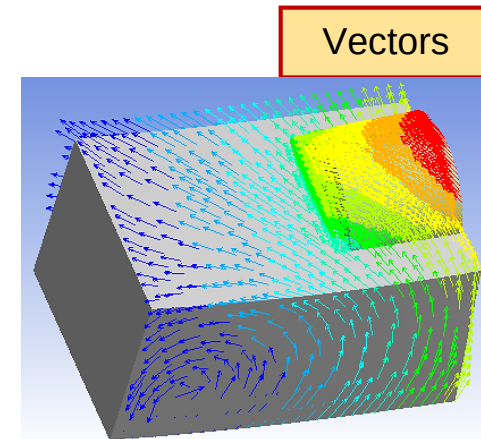
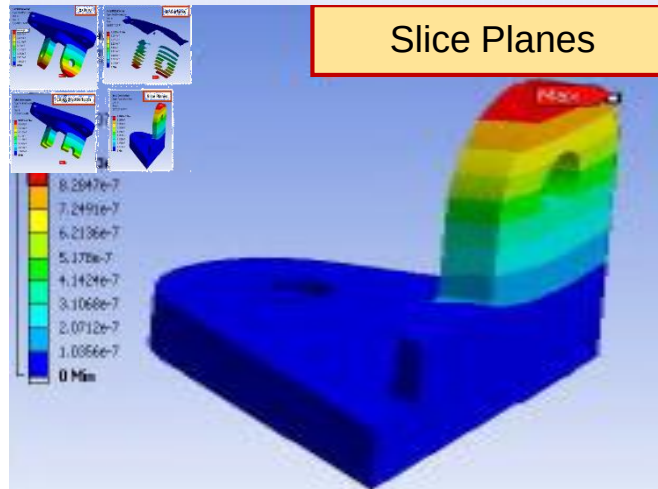
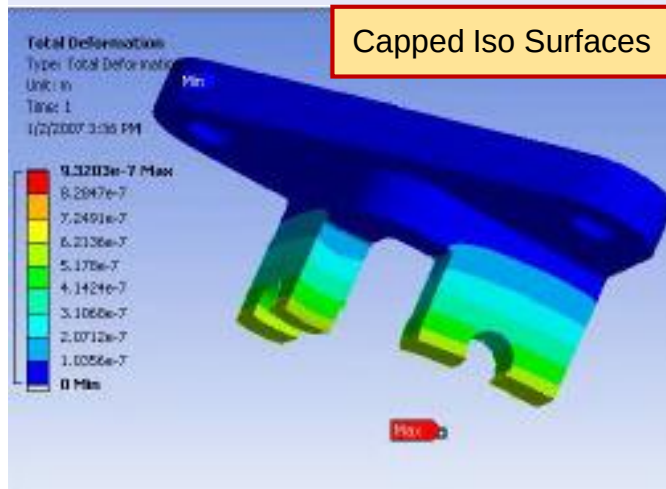
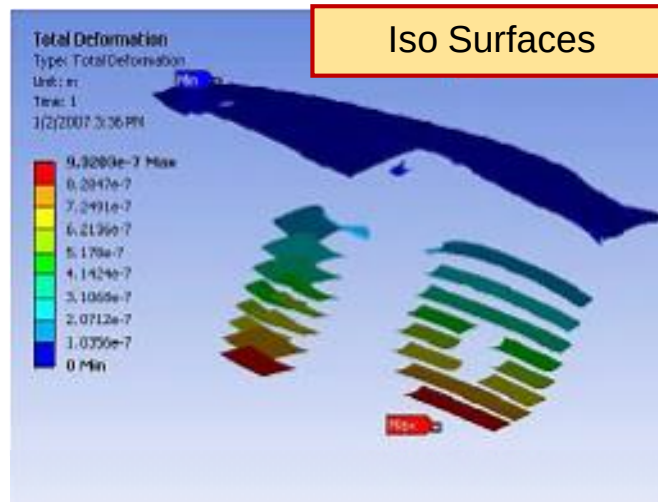
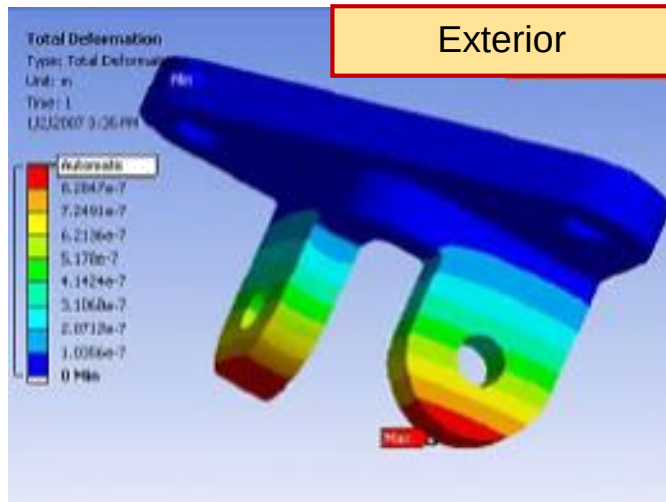


Introduction of ANSYS Workbench

■ 後處理(Post-processing)

➤ 結果顯示方式

	A
1	Static Structural
2	Engineering Data
3	DM Geometry
4	Model
5	Setup
6	Solution
7	Results



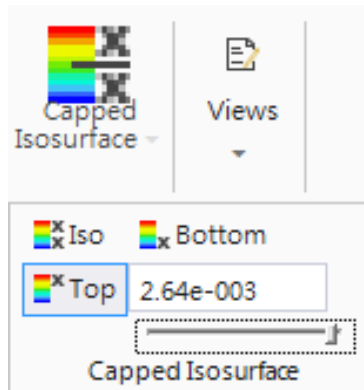
Introduction of ANSYS Workbench

A	
1	Static Structural
2	Engineering Data ✓
3	DM Geometry ✓
4	Model ✓
5	Setup ?
6	Solution ⚡
7	Results ⚡

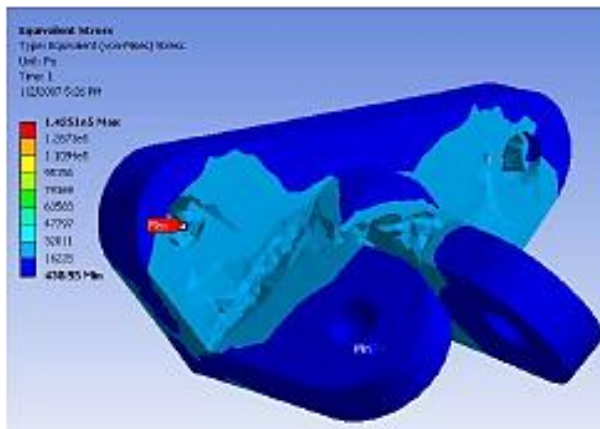
■ 後處理(Post-processing)

➤ Capped ISO surface

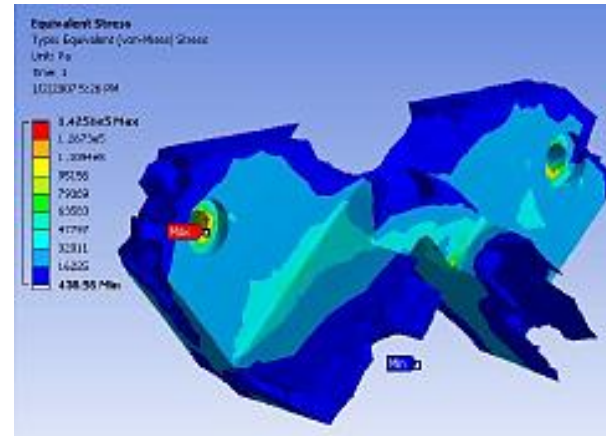
- ✓ 可設定閾值以外區域之圖案不顯示



Top capped : 超過閾值區域不顯示
Bottom capped : 低於閾值區域不顯示



頂部封頂等值面

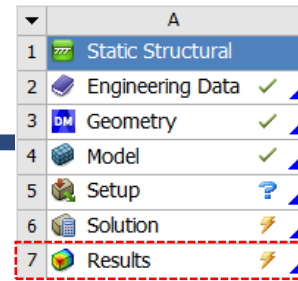


底部封頂等值面

Introduction of ANSYS Workbench

■ 後處理(Post-processing)

➤ 多觀察視窗



ANSYS 2019 R3 - Mechanical [ANSYS Mechanical Enterprise]

File Home Result **Display** Selection Automation

Isometric Views Next Rotate +Sx Rotate +Sz Rotate -Sy Pan Up Zoom In Pan Right Random Preferences Display Show Mesh Thick Shells and Beams Style Cross Section Display Style Show Vertices 0.12 (Auto Scale) Close Vertices Direction Thicken Color Explode Viewports Show Display

Outline

- Project*
- Model (A4)
 - Geometry
 - Solid
 - Materials
 - Coordinate Systems
 - Mesh
 - Automatic Method
 - Body Sizing
 - Static Structural (A5)
 - Analysis Settings
 - Force
 - Fixed Support
 - Solution (A6)
 - Solution Information
 - Total Deformation
 - Equivalent Stress

Mesh

Force

Total Deformation

Equivalent Stress

Details of "Total Deformation"

Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies

Definition	
Type	Total Deformation
By	Time
Display Time	Last
Calculate Time History	Yes
Identifier	
Suppressed	No

Results	
Minimum	0. mm
Maximum	2.6374e-003 mm
Average	3.672e-004 mm
Minimum Occurs On	Solid

Graph

Animation 20 Frames 2 Sec (Aut)

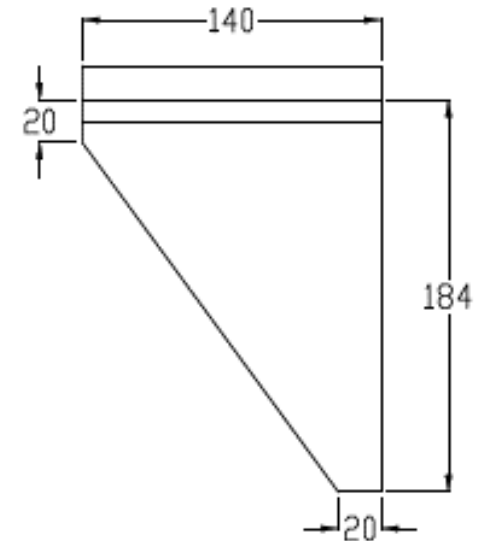
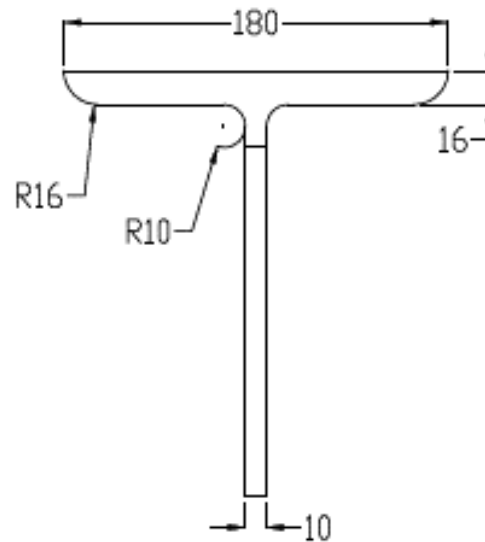
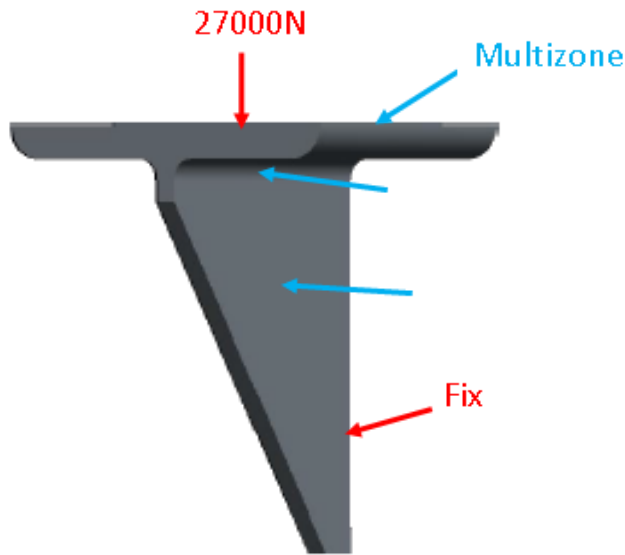
Tabular Data

Time [s]	Minimum [mm]	Maximum [mm]	Average [mm]
1.	0.	2.6374e-003	3.672e-004

Messages Graph

3D Solution – Ex.8 (來源：成功大學李輝煌教授)

機翼模型，尺寸如下所示，使用MultiZone之網格方法將翼板頂面、腹板側面、接合處圓角面進行mesh設定，並將翼板與腹板接合處之圓角兩面設定element size為7的mesh。邊界條件如圖所示，板子後方之面固定，上方施予頂面一力。觀察其等效應力、位移量變化、結構誤差及Safety Factor。材料選用鋼。(單位：MM, N)



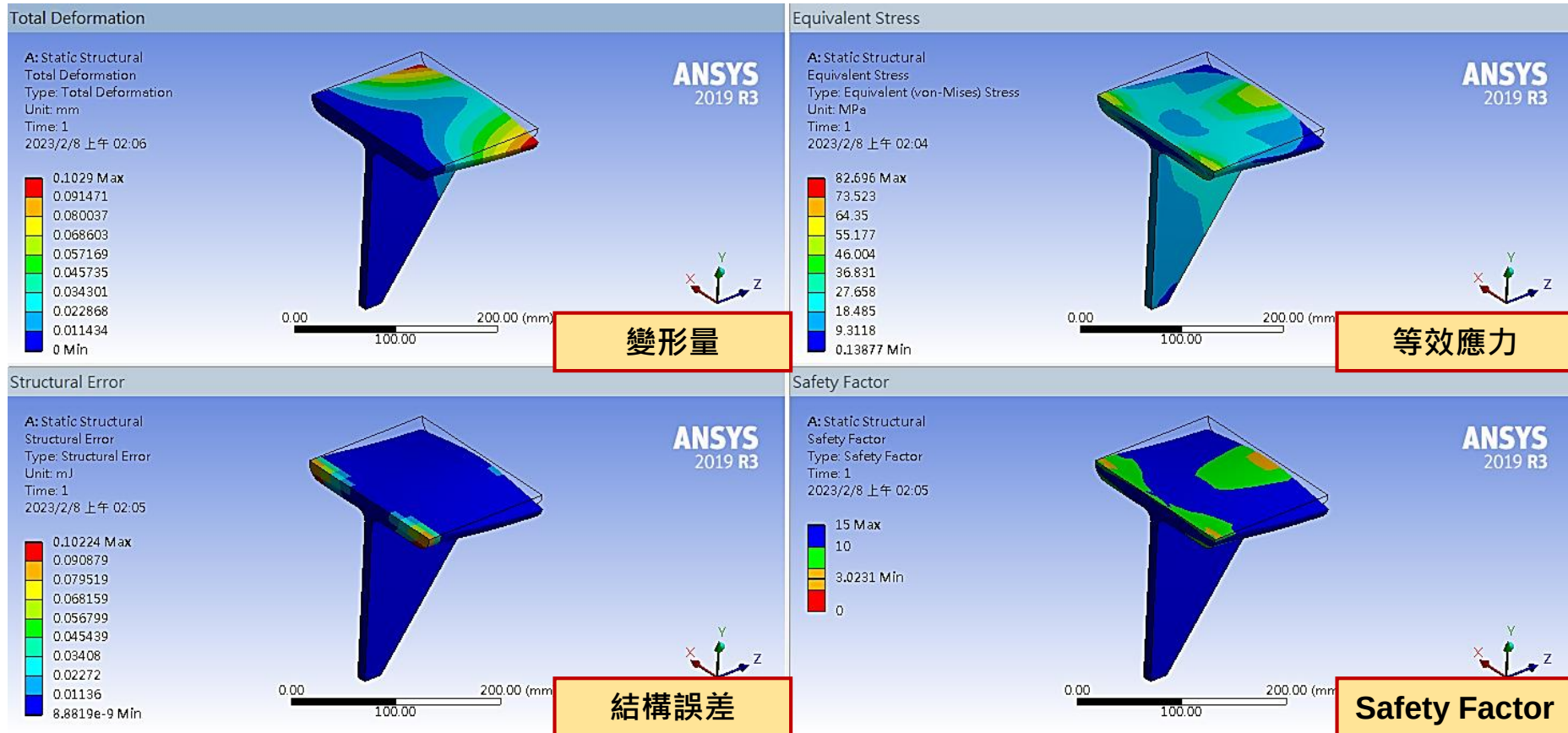
3D Solution – Ex.8

學習目標

- Blend
- Mesh-MultiZone
- 後處理顯示

機翼模型，尺寸如下所示，使用MultiZone之網格方法將翼板頂面、腹板側面、接合處圓角面進行mesh設定，並將翼板與腹板接合處之圓角兩面設定element size為7的mesh。邊界條件如圖所示，板子後方之面固定，上方施予頂面一力。觀察其等效應力、位移量變化、結構誤差及Safety Factor。材料選用鋼。(單位：MM, N)

多視窗顯示



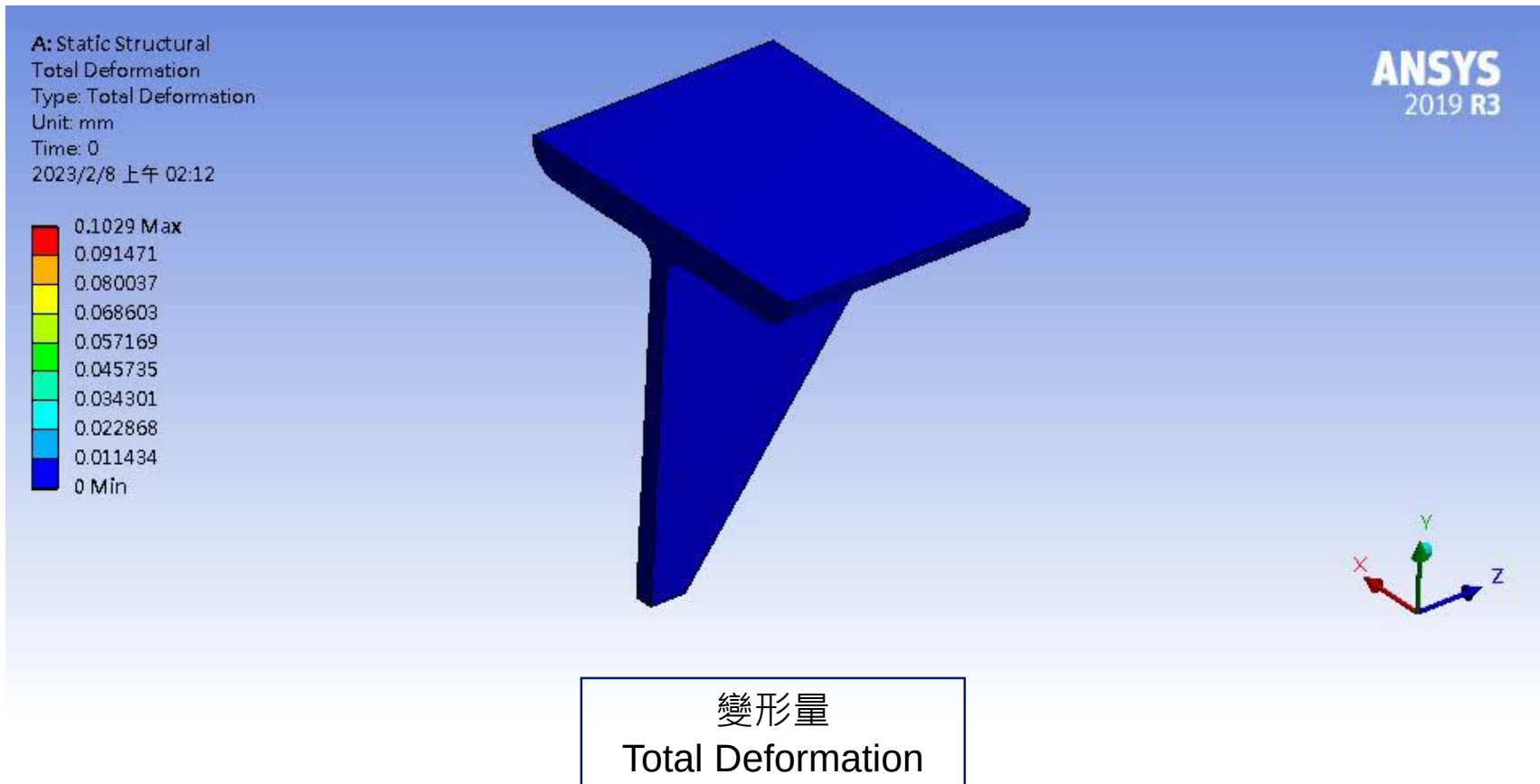
3D Solution – Ex.8

學習目標

- Blend
- Mesh-*MultiZone*
- 後處理顯示

機翼模型，尺寸如下所示，使用MultiZone之網格方法將翼板頂面、腹板側面、接合處圓角面進行mesh設定，並將翼板與腹板接合處之圓角兩面設定element size為7的mesh。邊界條件如圖所示，板子後方之面固定，上方施予頂面一力。觀察其等效應力、位移量變化、結構誤差及Safety Factor。材料選用鋼。(單位：MM, N)

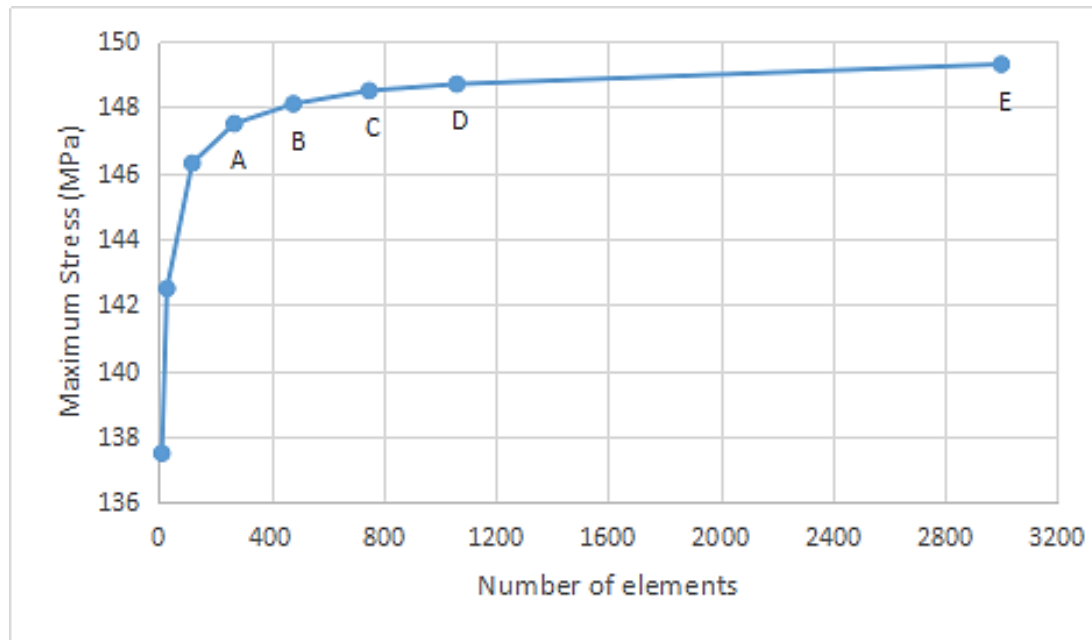
動畫顯示





Convergence

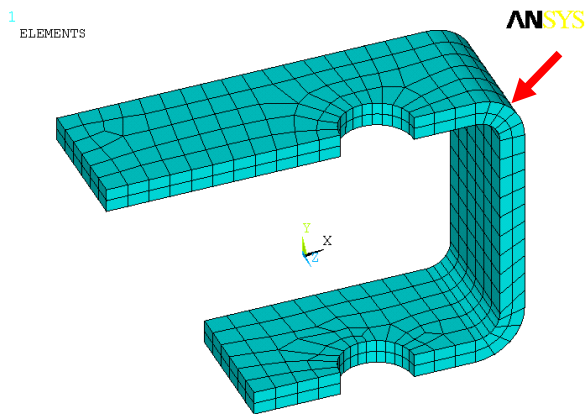
- 隨著網格密度增加(即元素增加、mesh尺寸減小)，有限元素分析所求出的解會趨近於一個定值，即為該題目的正確解(exact solution)，但隨著網格變得更精細，所消耗的運算資源也會增加
- 收斂性分析：當進一步細分網格後所求得解變化很小時，即可認為網格已經收斂了
- 收斂物理量通常可為應力/位移/能量，誤差最好於5%內



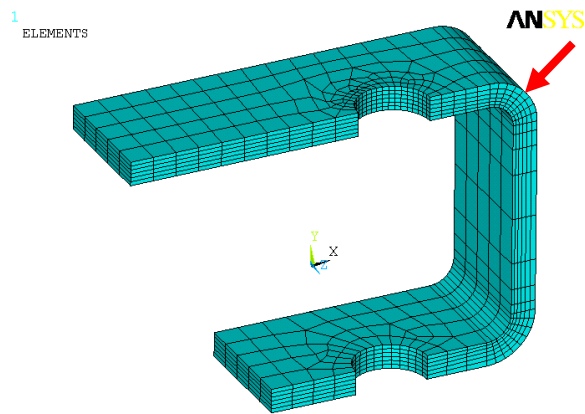
Convergence



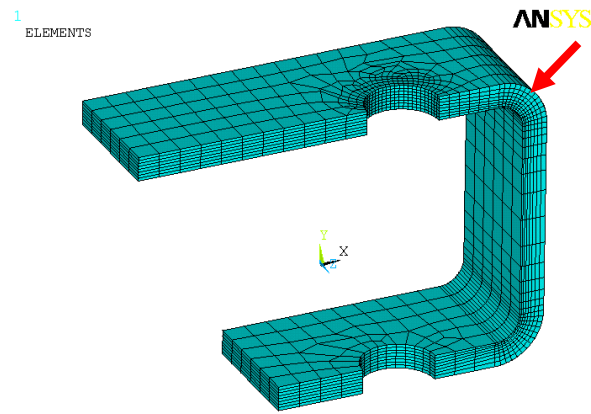
圓角處Mesh元素大小



5.00 mm

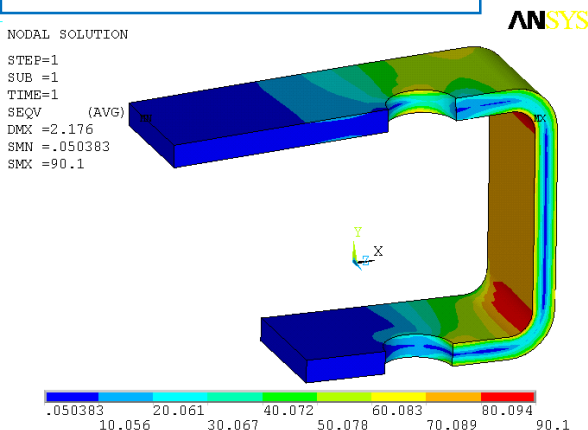


1.15 mm

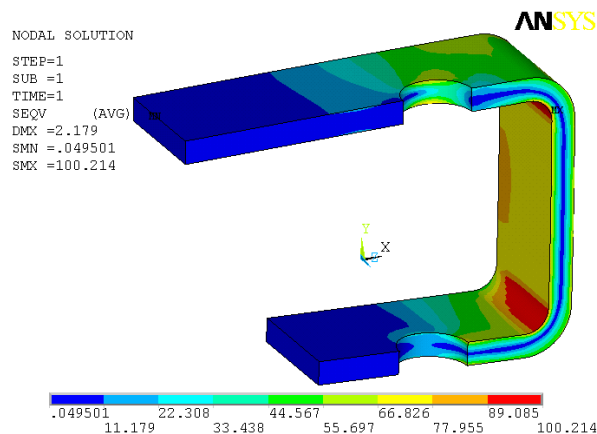


0.67 mm

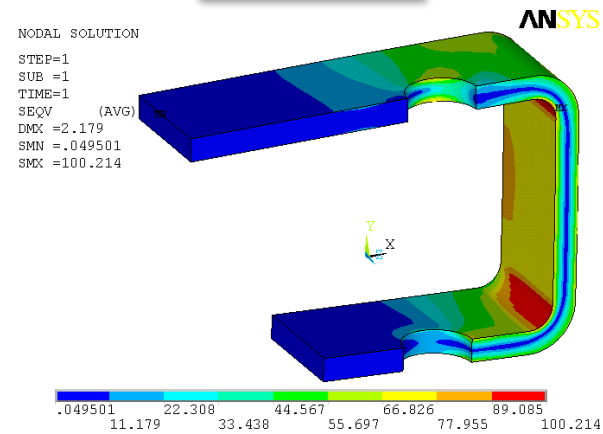
圓角處最大等效應力值



90.100 MPa



100.214 MPa



100.333 MPa

模型收斂

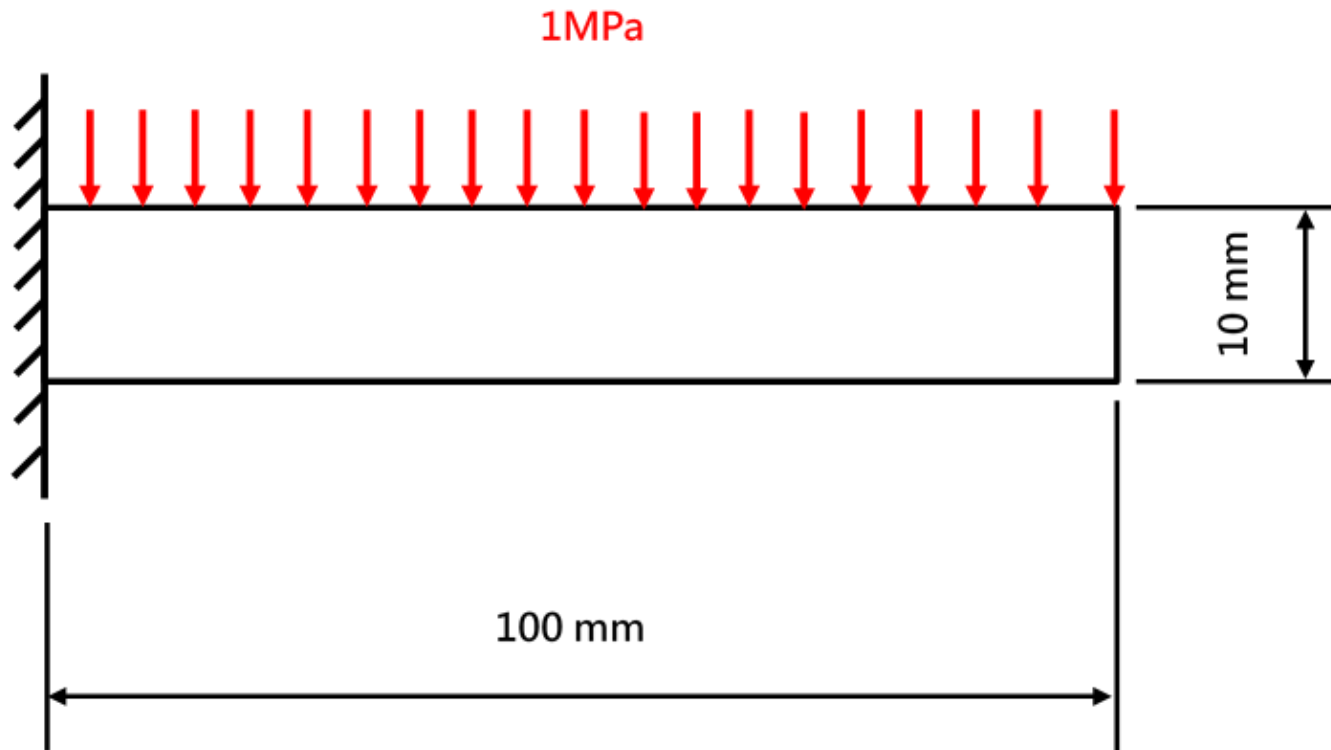
11.2%

0.12%



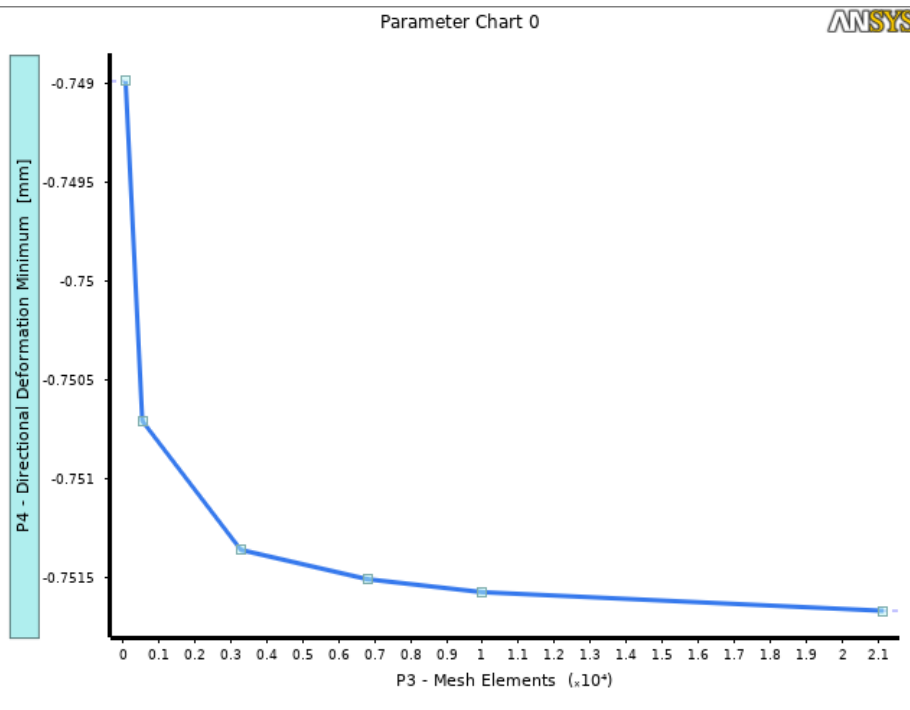
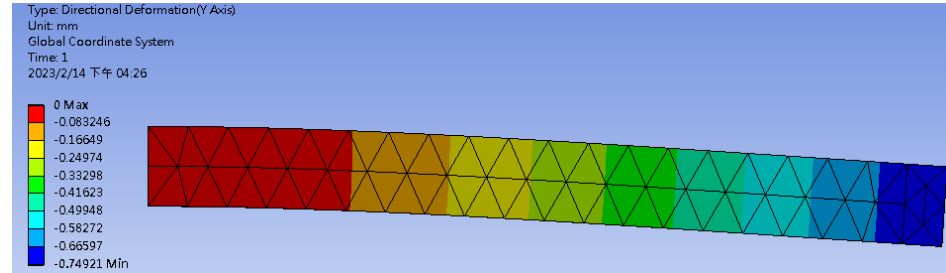
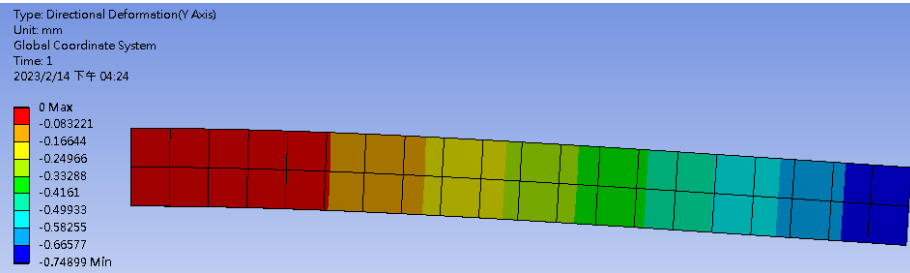
Convergence – Ex.9 (來源：成功大學李輝煌教授)

一材料為鋼(steel)製成之懸臂樑，尺寸為100x10x10mm，上端平面施以1MPa均佈負載，請應用不同元素大小(element size)探討懸臂樑模型之收斂性(1)Hex mesh、(2)Tet mesh

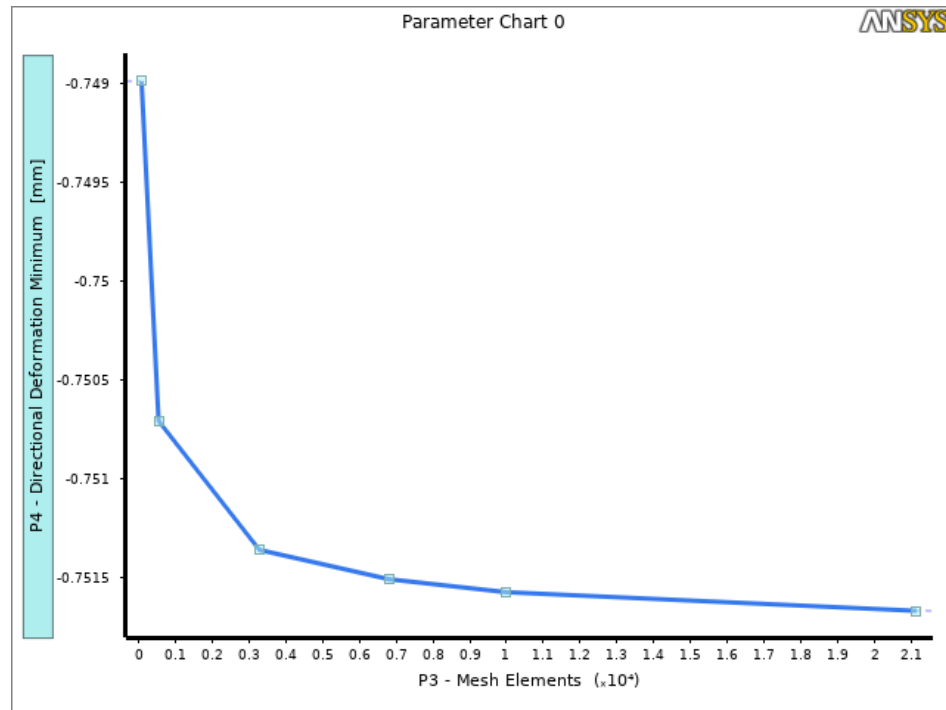


Convergence – Ex.9

一材料為鋼(steel)製成之懸臂樑，尺寸為100x10x10mm，上端平面施以1MPa均佈負載，請應用不同元素大小(element size)探討懸臂樑模型之收斂性(1)Hex mesh、(2)Tet mesh



Hex mesh



Tet mesh