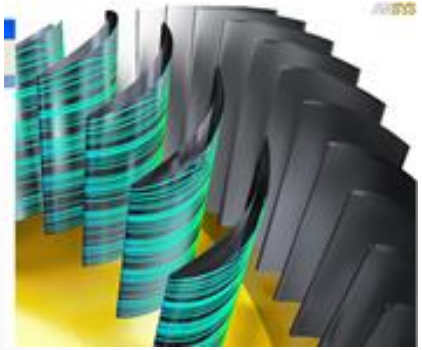
電腦輔助工程分析
ANSYS WORKBENCH

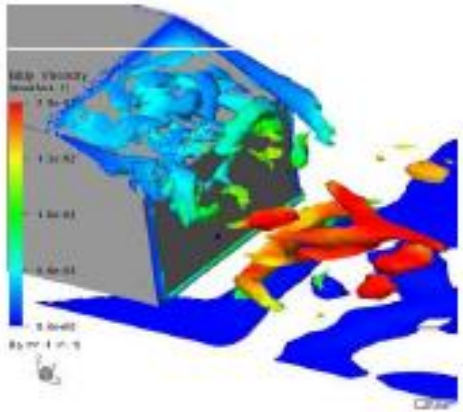
林峻立 教授 國立陽明大學 生物醫學工程系

ANSYS WORKBENCH

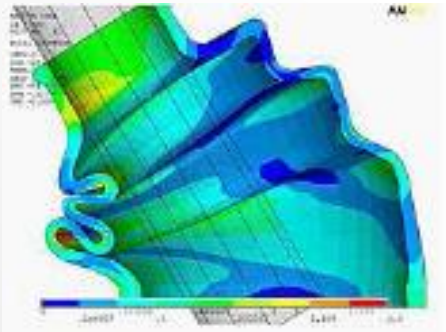
- Workbench 為開發用來提供一個強大與獨特的之模擬分析軟體。並提供一參數化及人性化界面供大部分使用者容易使用。

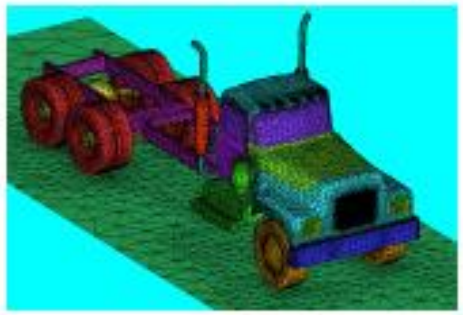
- 優點
  - 模型建構能力佳
  - 與CAD軟體結合及通用性高
  - 建模形之運算及網格切割能力佳
  - 結果圖案美觀效果佳



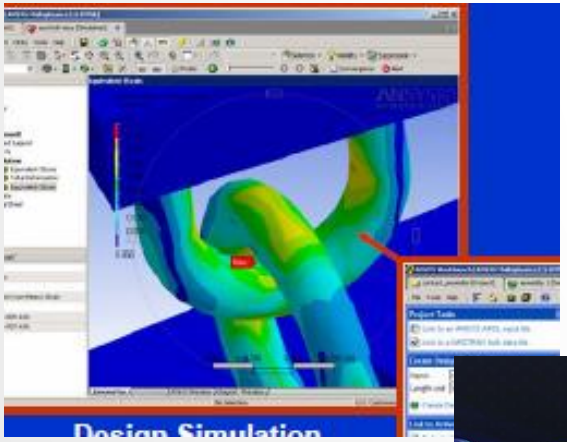


- 缺點
  - 過多數值被預設設定
  - 容易造成分析結果不正確
  - 結果觀察較難深入

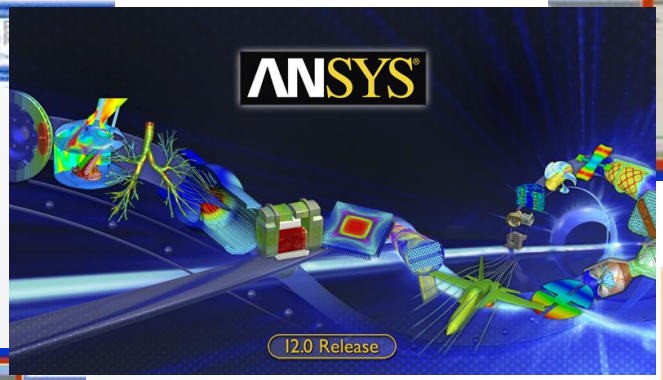


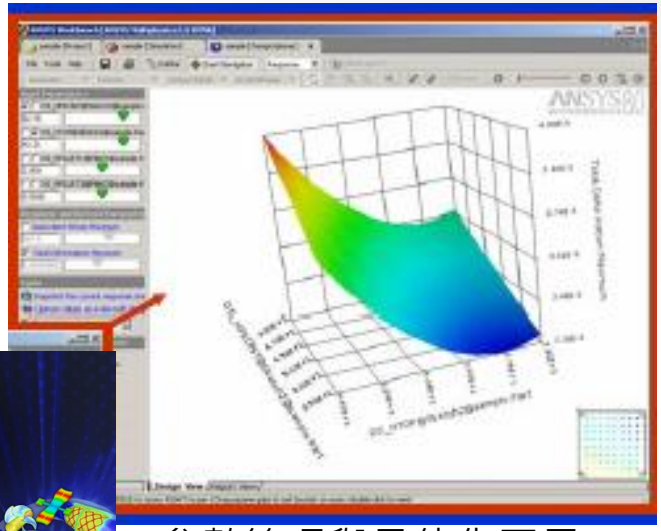


Workbench概述

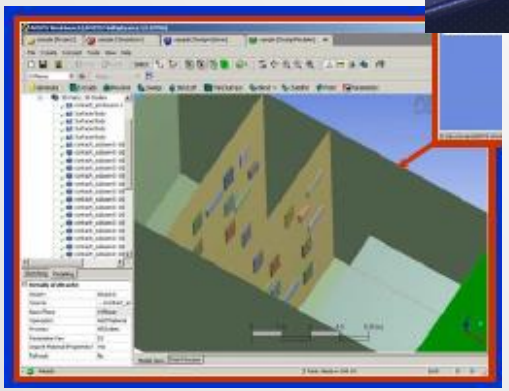


模擬分析
（DesignSpace）

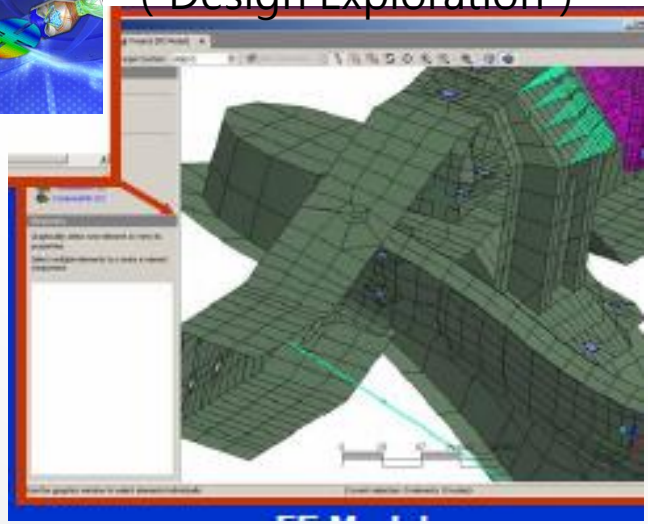




參數管理與最佳化工具
（Design Exploration）



參數化建模
（DesignModeler）

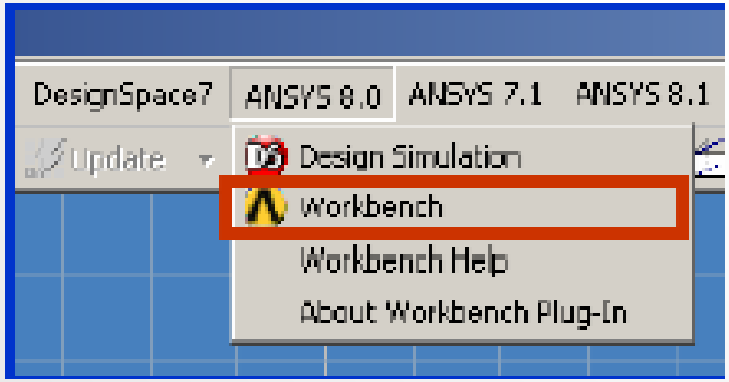


分析連結（FE Modeler）

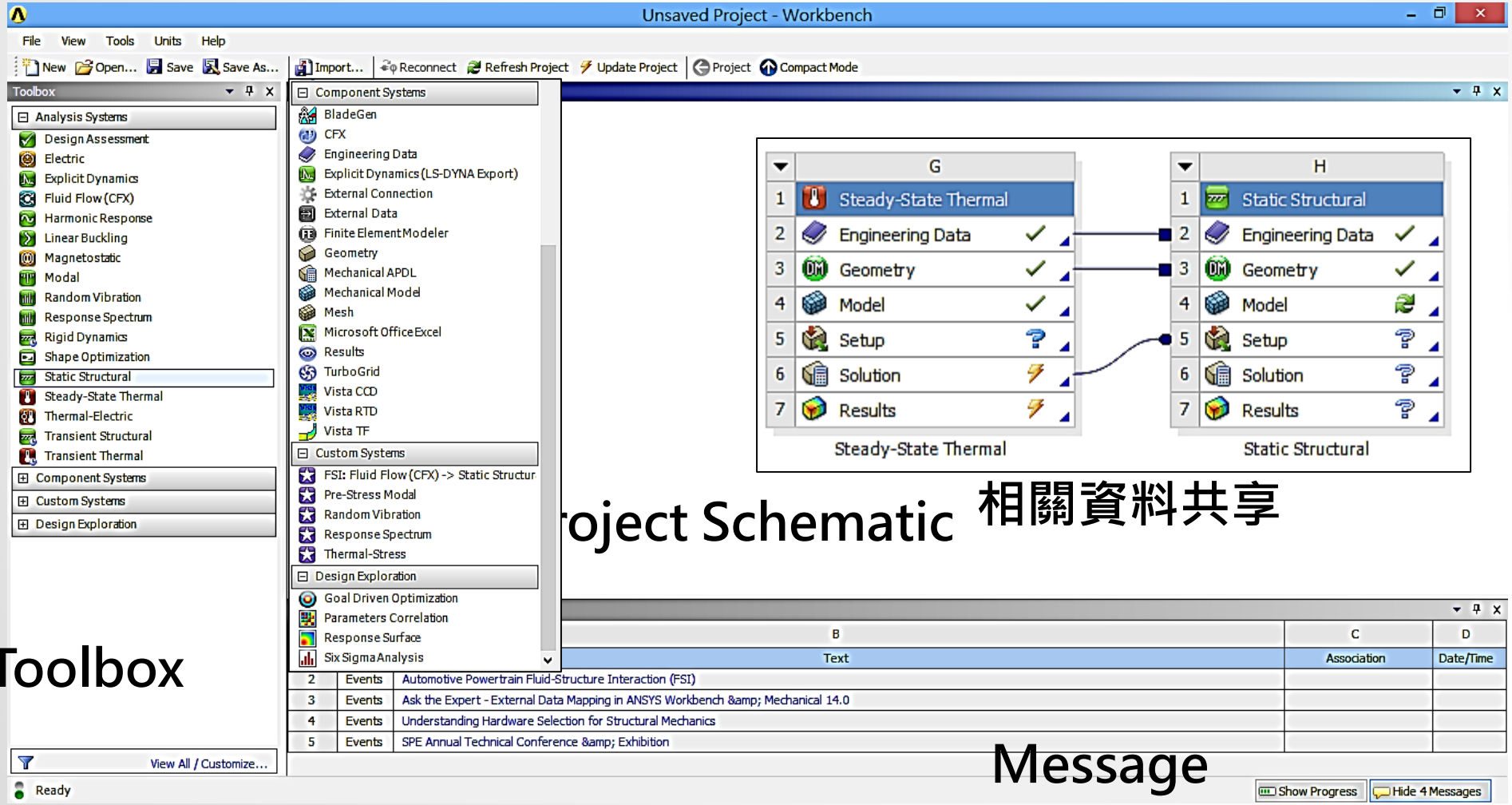
啟動ANSYS WORKBENCH

- 與ANSYS同一個開啟目錄

- 或者可由其他市面上CAD軟體系統中插入



起始界面



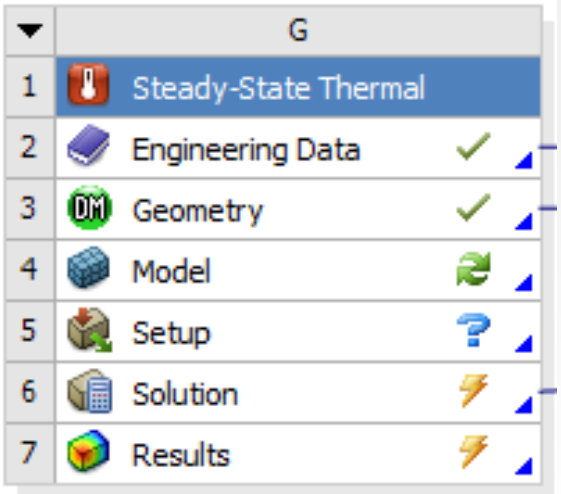
roject Schematic

相關資料共享

Toolbox

Message

狀態顯示



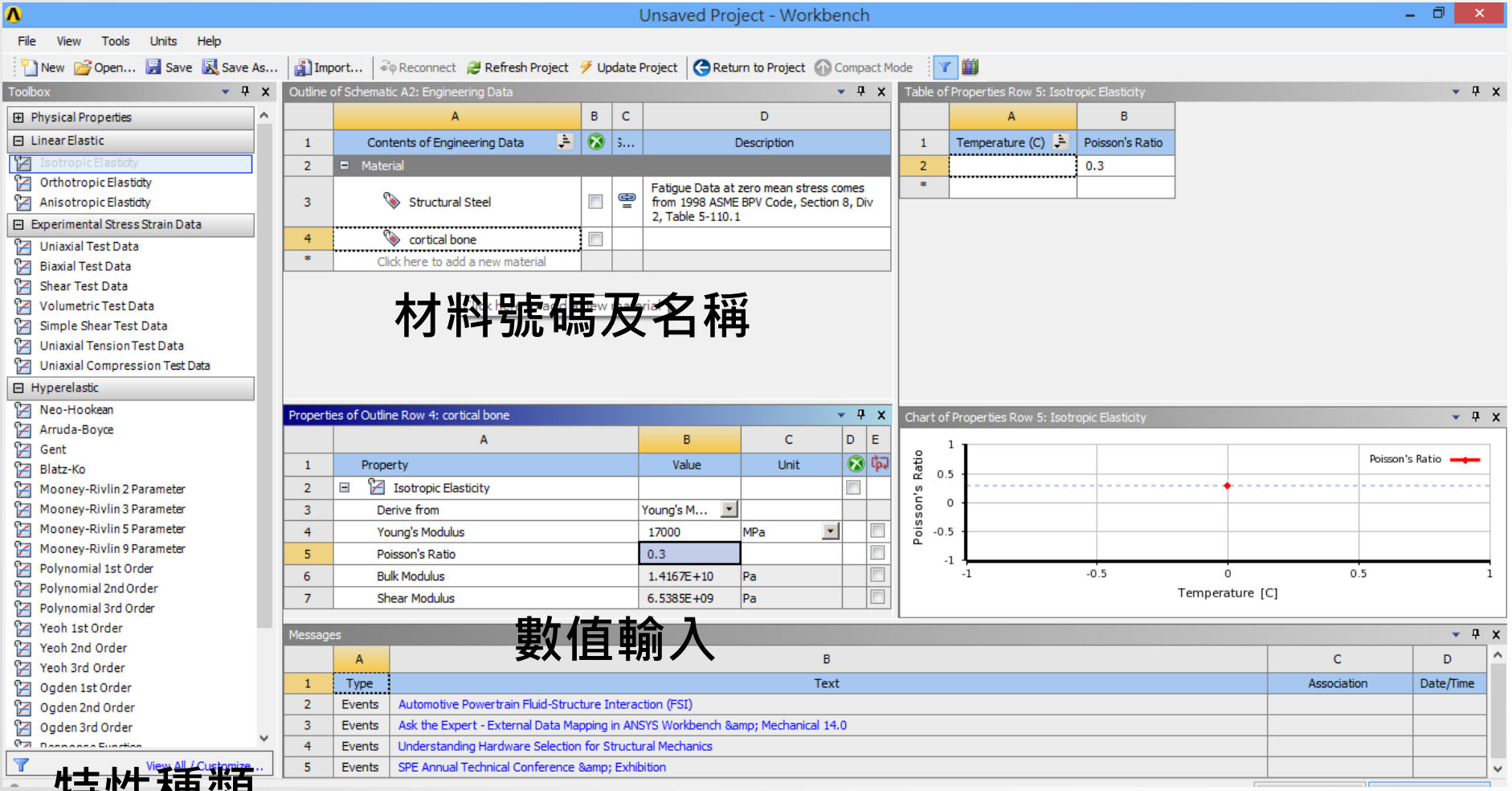
最新的狀態(數據輸入完整)

需要刷新：上游部數據已改變。需更新單元

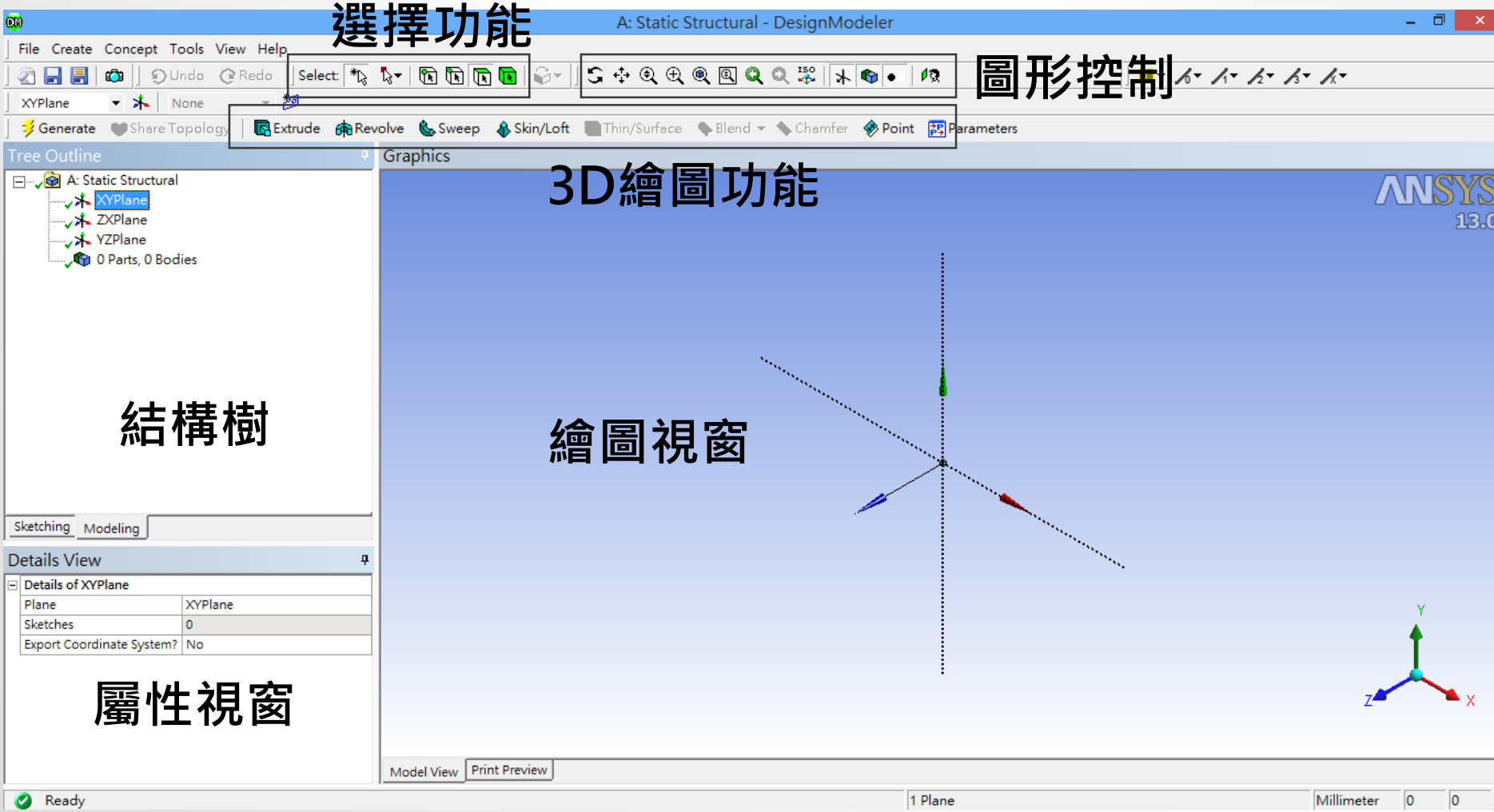
需要注意：可能需要改正本項資訊或是上游資訊

需要更新：數據已改變，輸出需重新產生

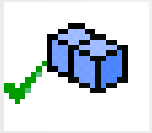
工程資料(Engineering Data)



Design Modeler



結構樹狀態顯示



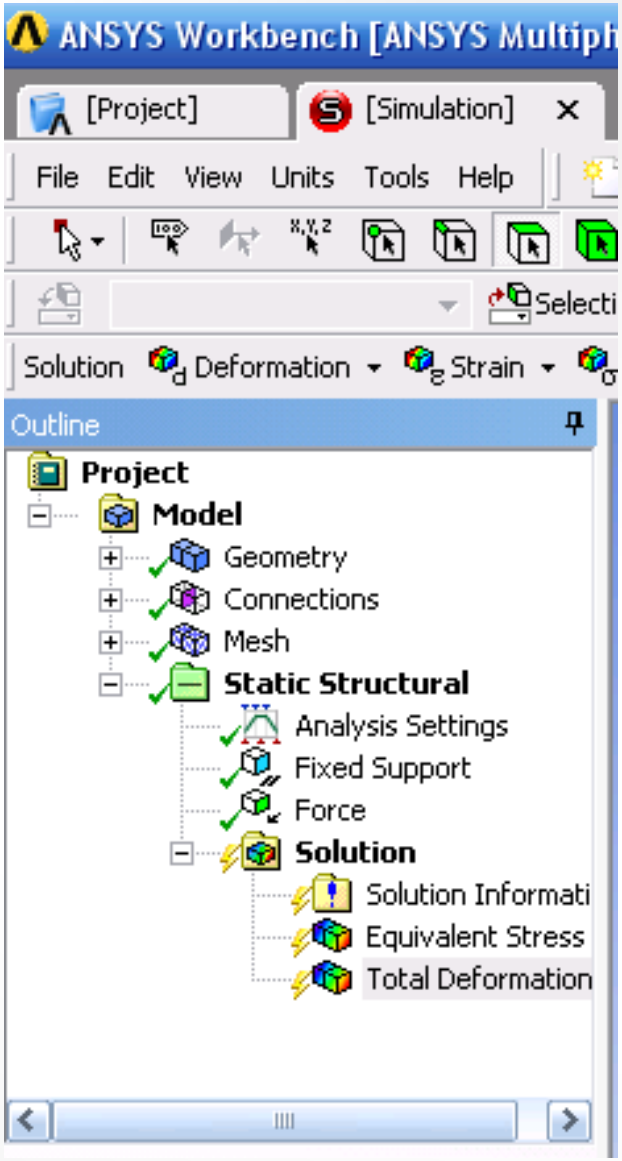
- 說明分支全部被定義

- 說明輸入的數據不完整

- 說明需要求解

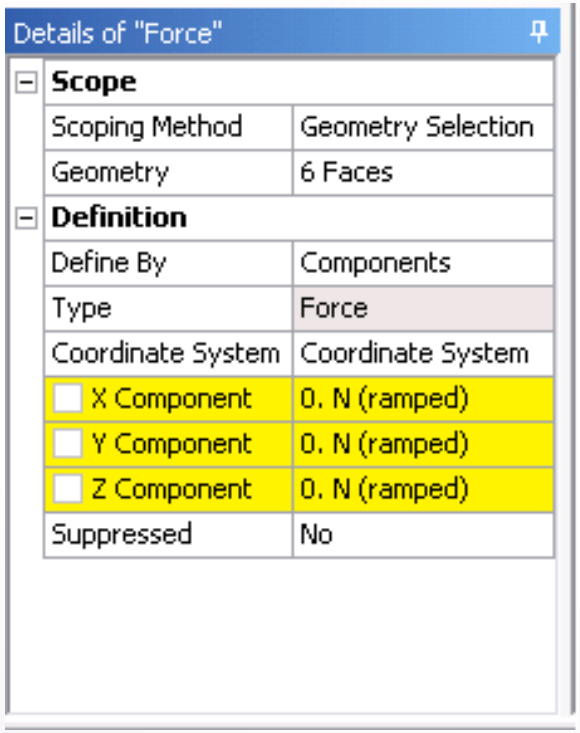
- 說明被抑制，不能被求解

- 說明體積或零件被隱藏



屬性視窗顯示

- 白色區域：顯示當前輸入的數據。(可編輯)
- 灰色區域：顯示信息數據。(不可編輯)
- 黃色區域：未完成的信息輸入

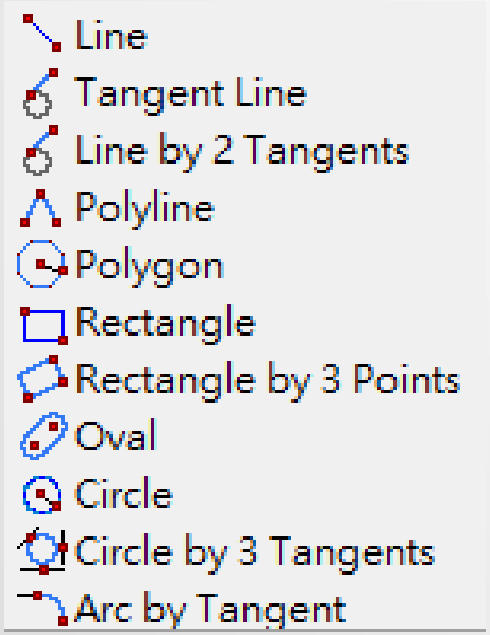


Design Modeler

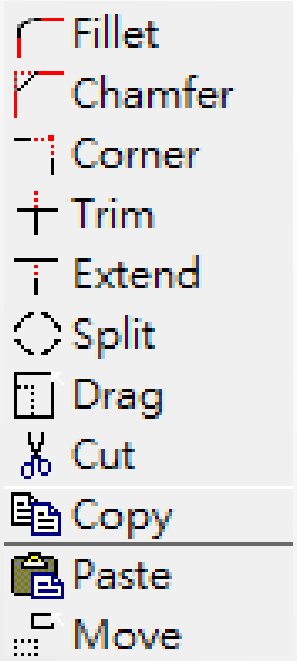
- 草圖模式
  - 包括建構二維幾何模型。此二維幾何模型可作為3D模型建構之依據。
- 3D建模
  - 將草圖進行拉伸旋轉等操作建構3D幾何模型。
- CAD模型輸入
  - 直接導入商業化CAD模型進入，並對其進行修正。
- 概念建模
  - 用於創建和修改直線和表面實體，使之能用於代表樑和殼之有限元素模型。

2D Sketching

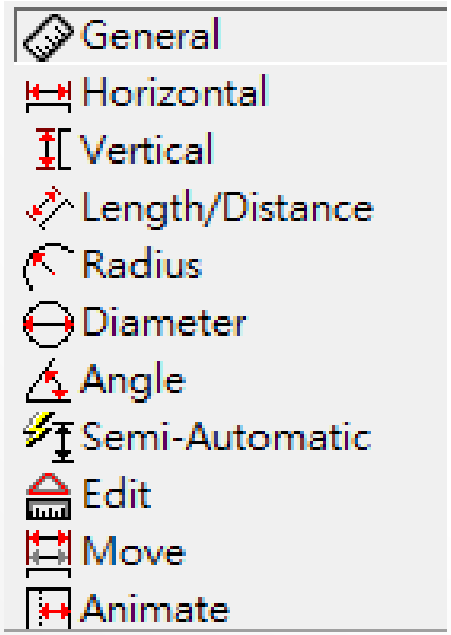
Draw



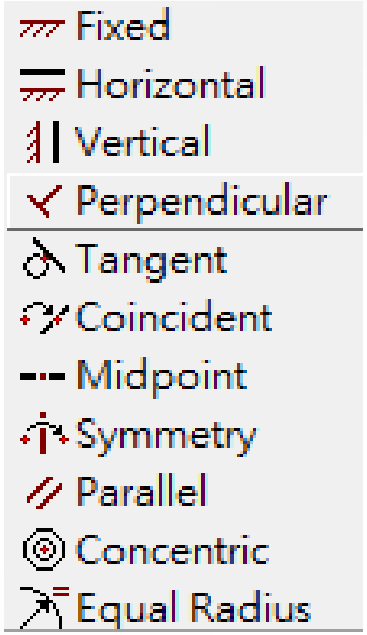
Modify



Dimensions

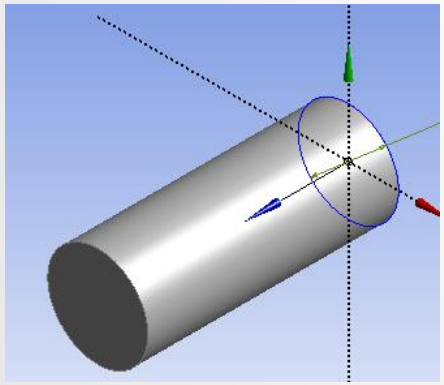


Constraints

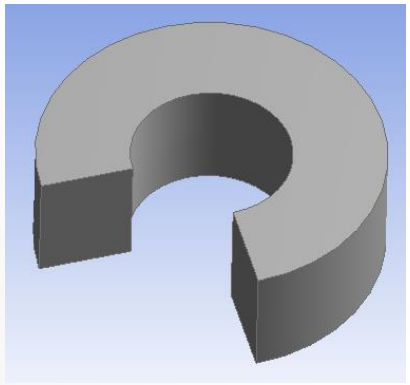


3D Modeling

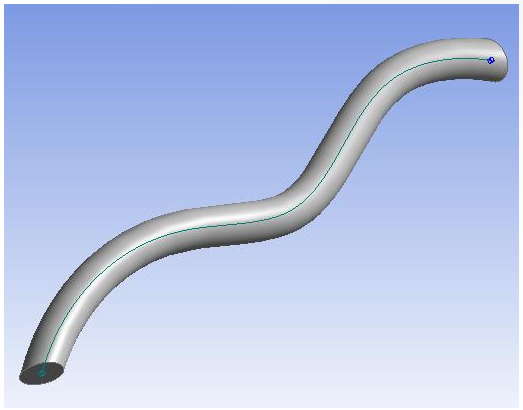
Extrude



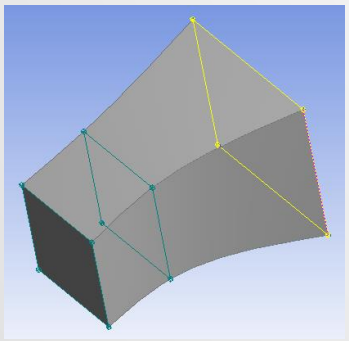
Revolve



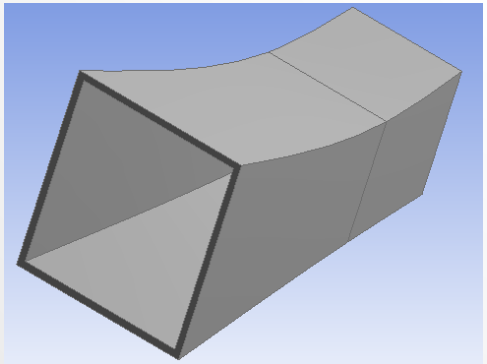
Sweep



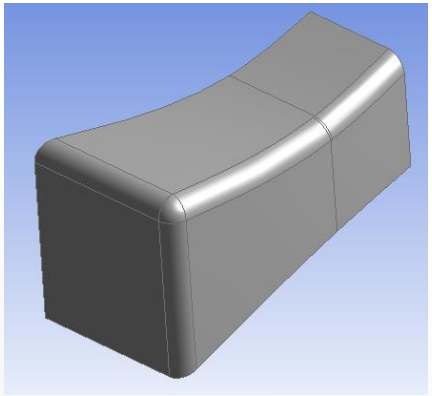
Skin/Loft



Thin/Surface

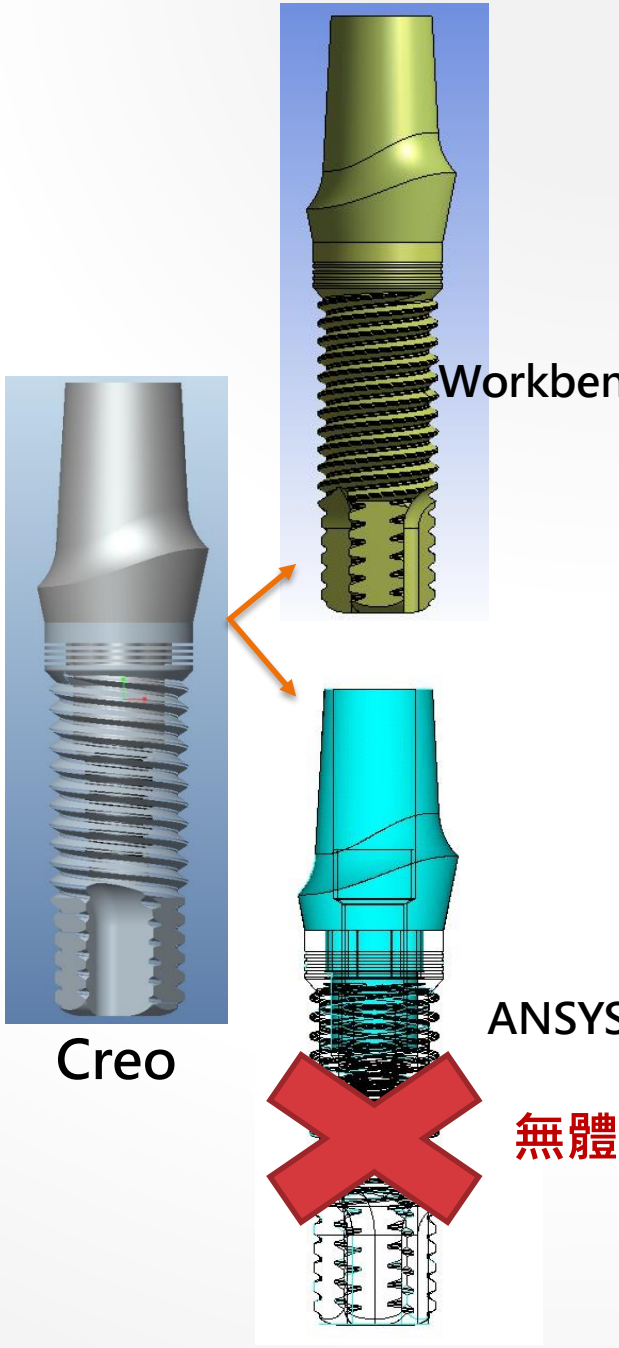


Blend

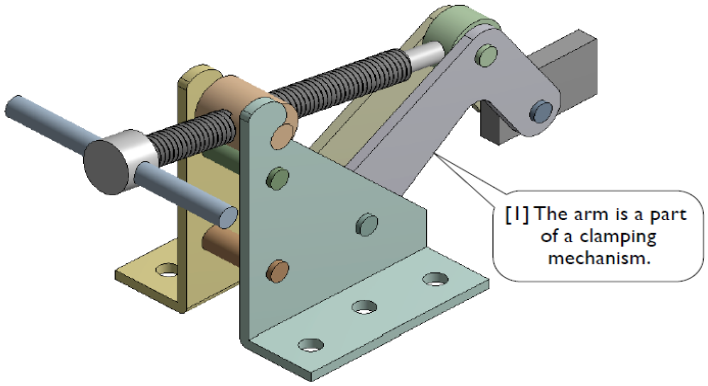


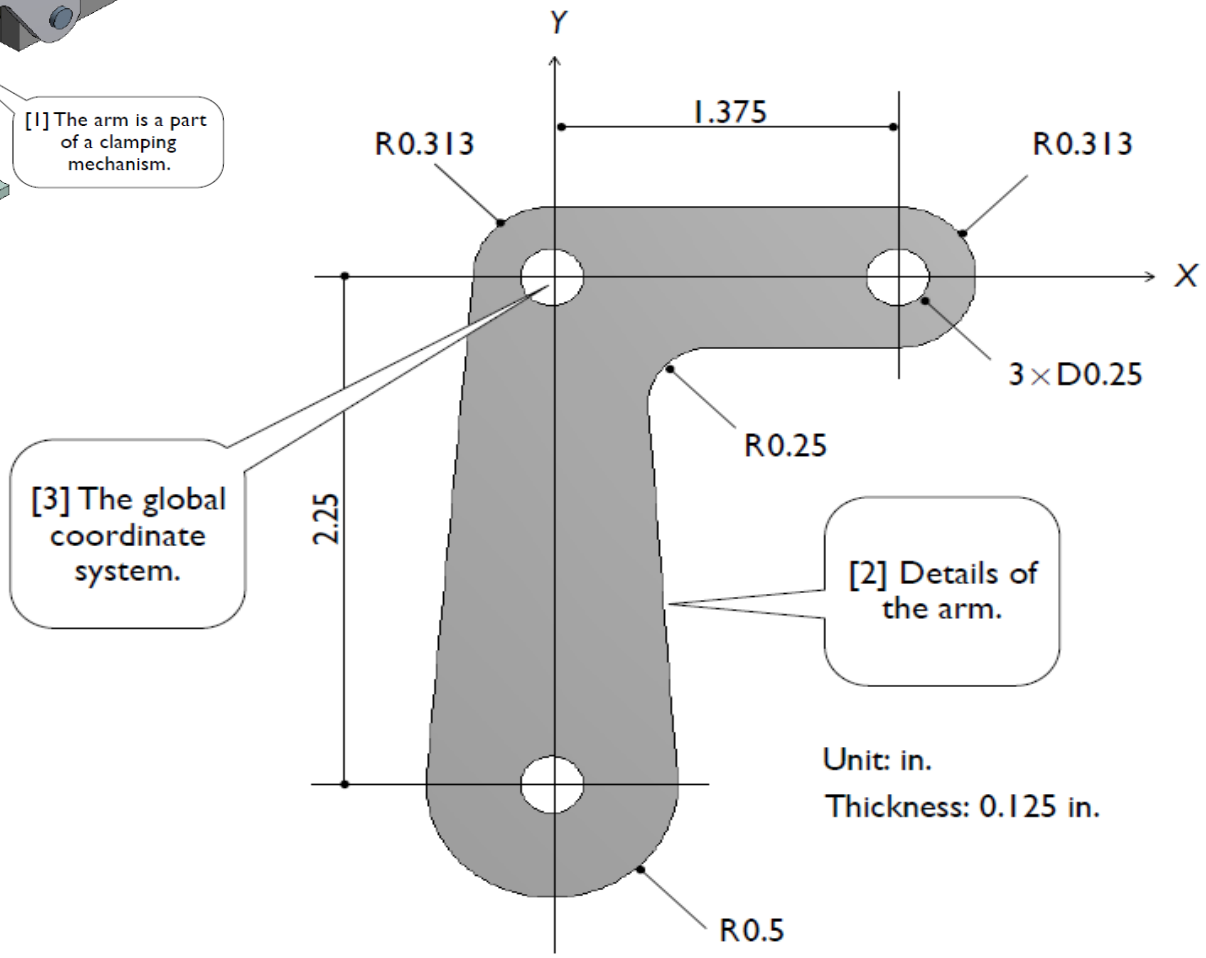
CAD模型輸入

- 3D Modeling
  - Import External Geometry File
    - UG NX(*.prt)，CATIA(*.model, *.CATpart)
    - Creo(*.prt, *.asm)，Solid Edge(*.par,*.asm)
    - SolidWorks(*.sldprt, *.sldasm)，STEP(*.stp)
    - ….
  - Attach to Active CAD Geometry
    - CATIA
    - Creo（Pro/ENGINEER）
    - SolidWorks
    - Solid Edge
    - …



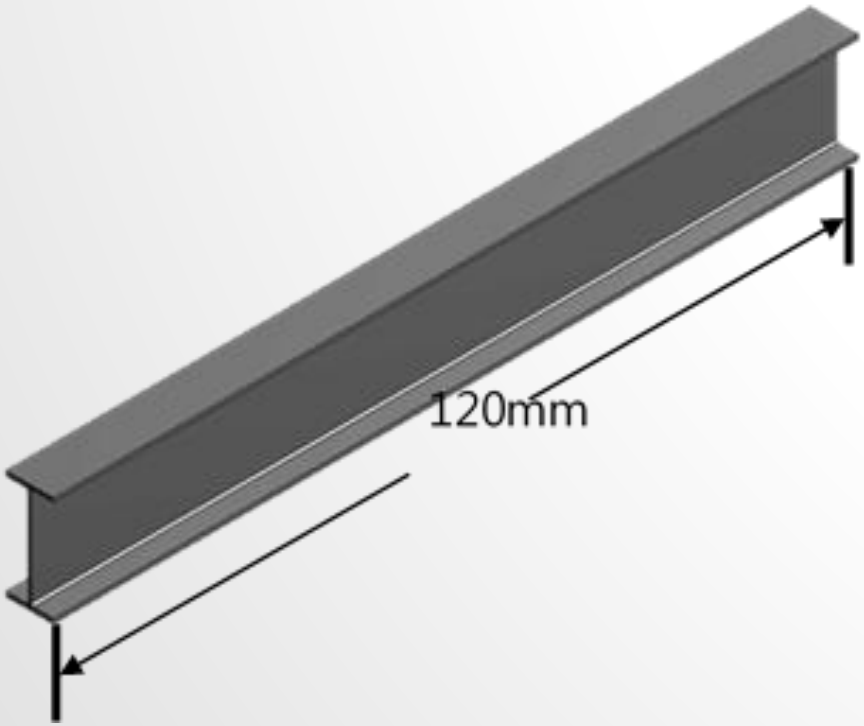
Exercise 1 CAD-1 (來源：成功大學李輝煌教授)

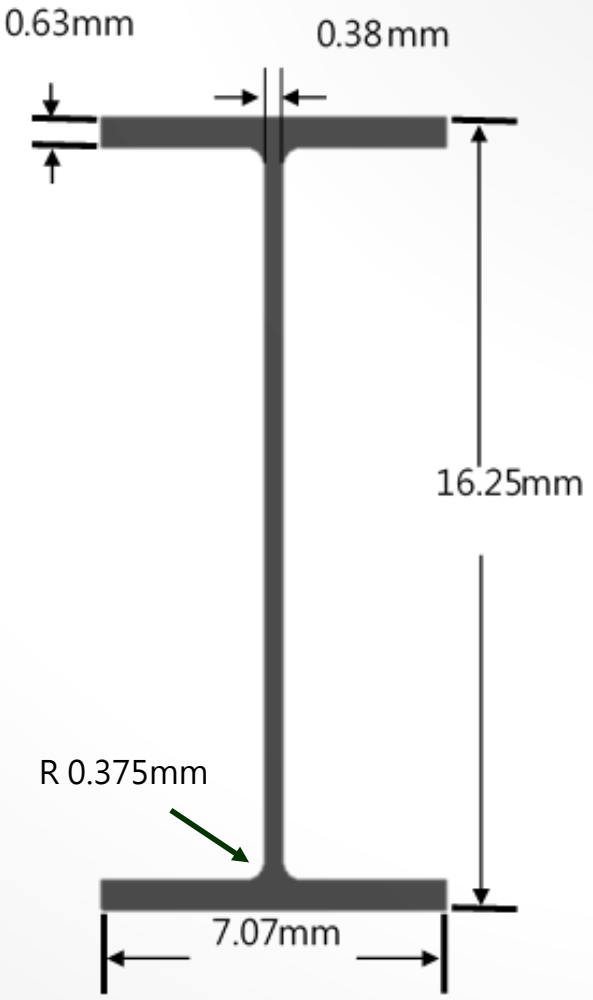




Exercise 2 CAD-2 (來源：成功大學李輝煌教授)

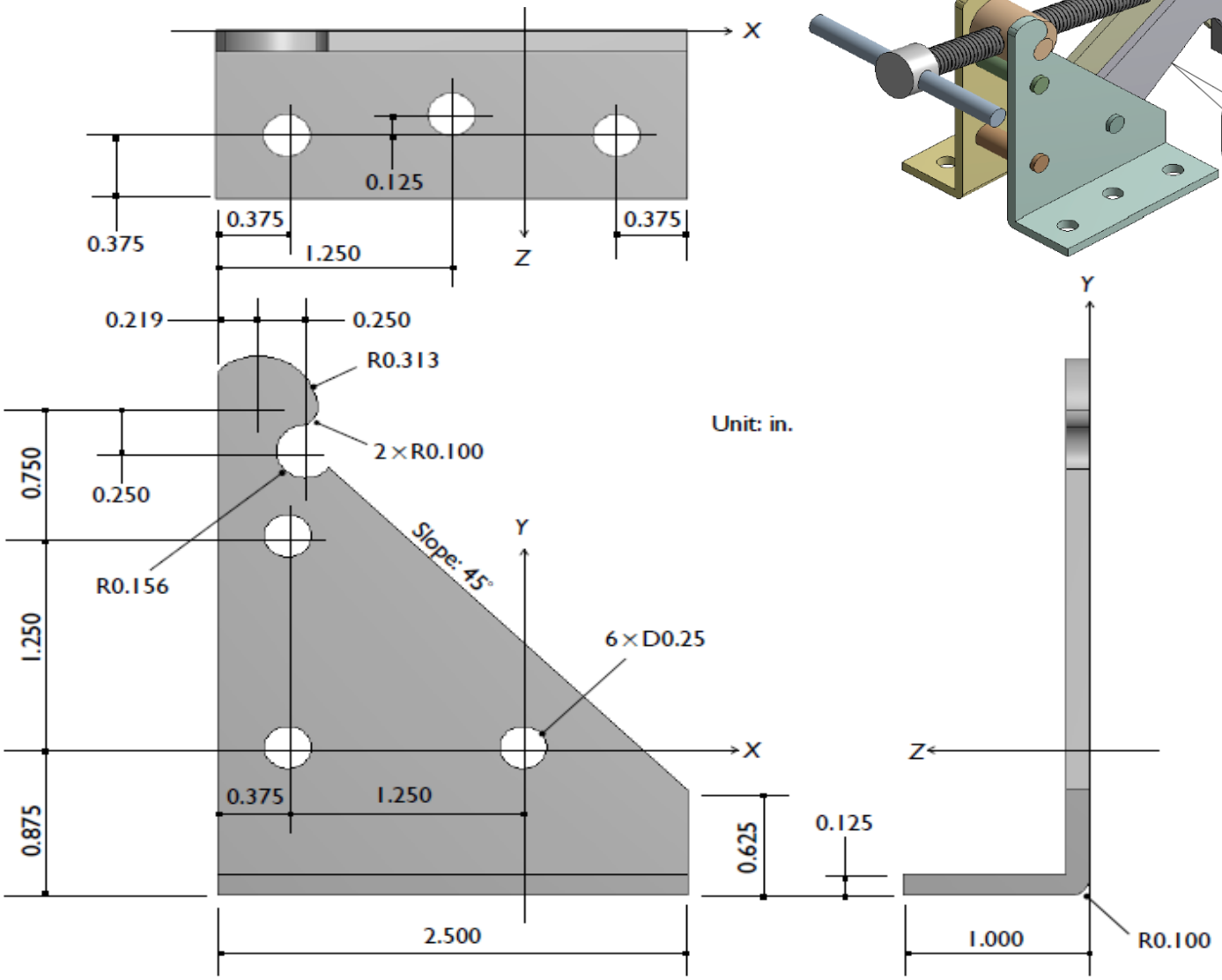
- I Beam model

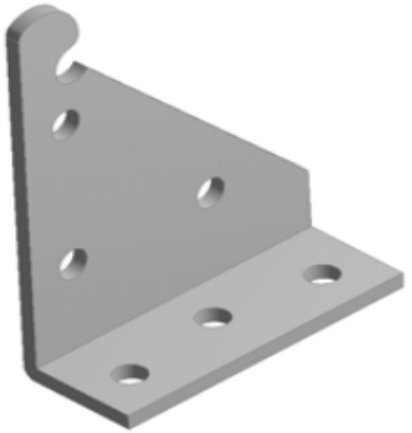




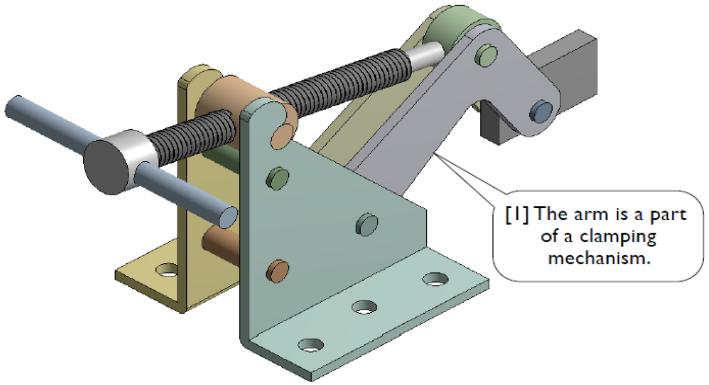
Cross section

Exercise 3 CAD-3 (來源：成功大學李輝煌教授)

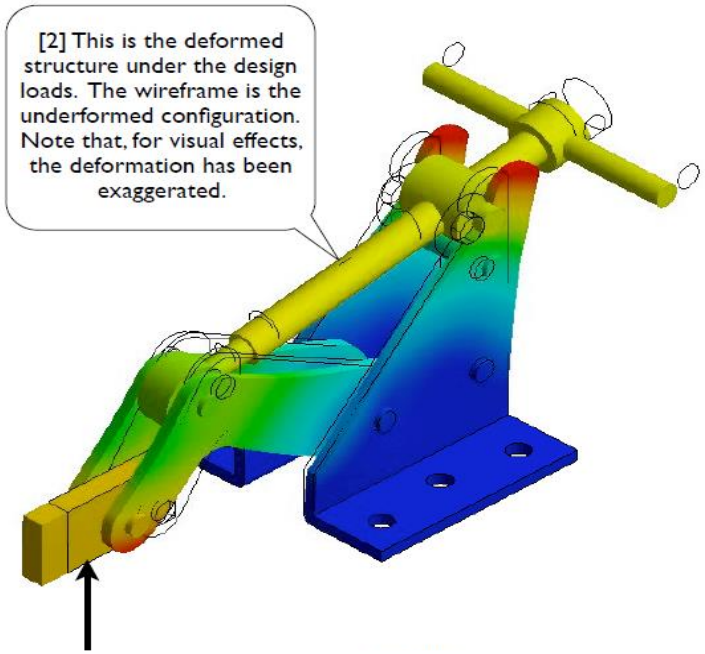




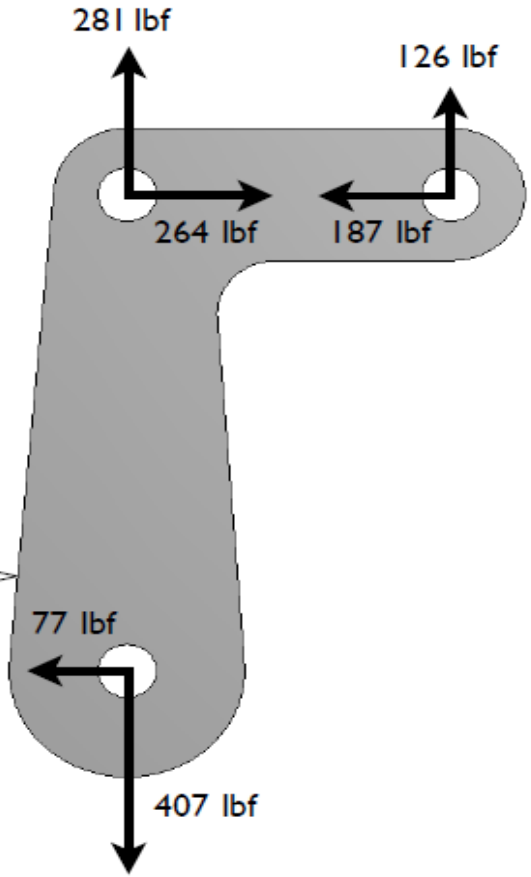
Exercise 4 CAE-1 (來源：成功大學李輝煌教授)



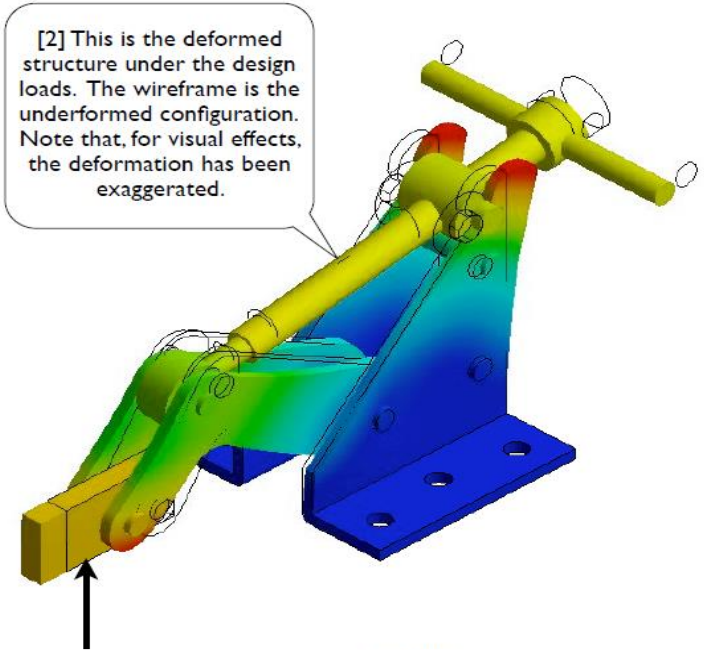
[1] The clamping mechanism is designed to withstand a clamping force of 450 lbf.

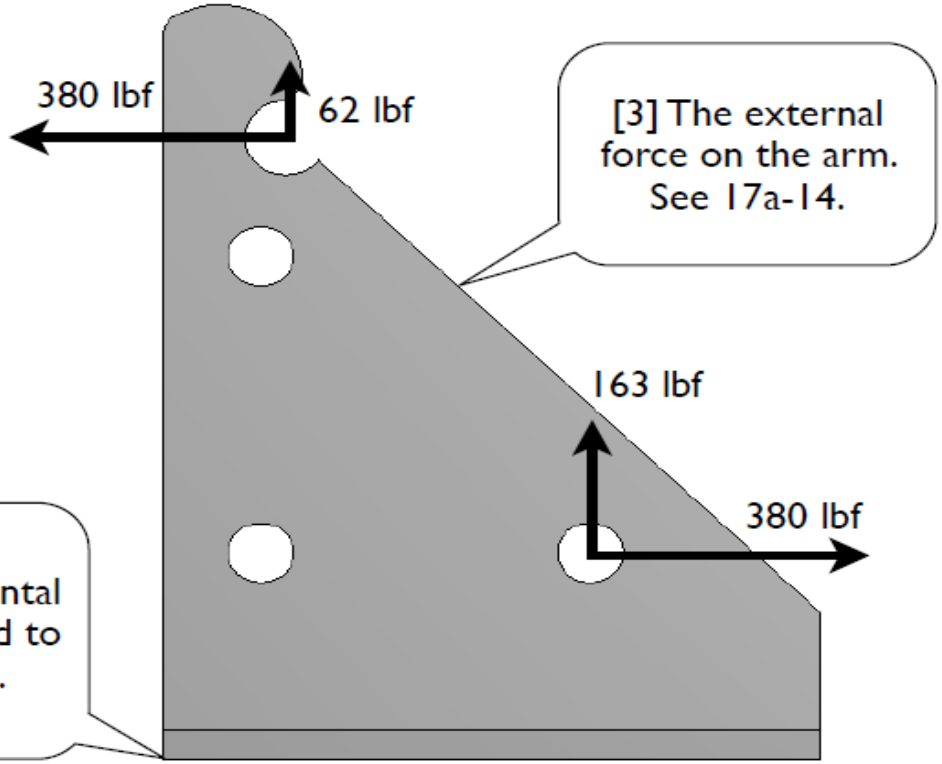


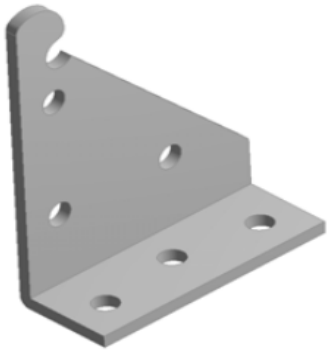
[3] The external forces on the arm. These forces are calculated according to 17a-13.



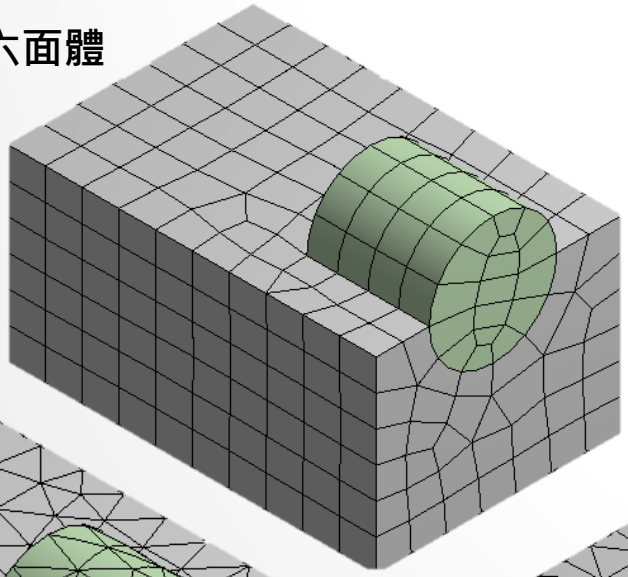
Exercise 5 CAE-2 (來源：成功大學李輝煌教授)



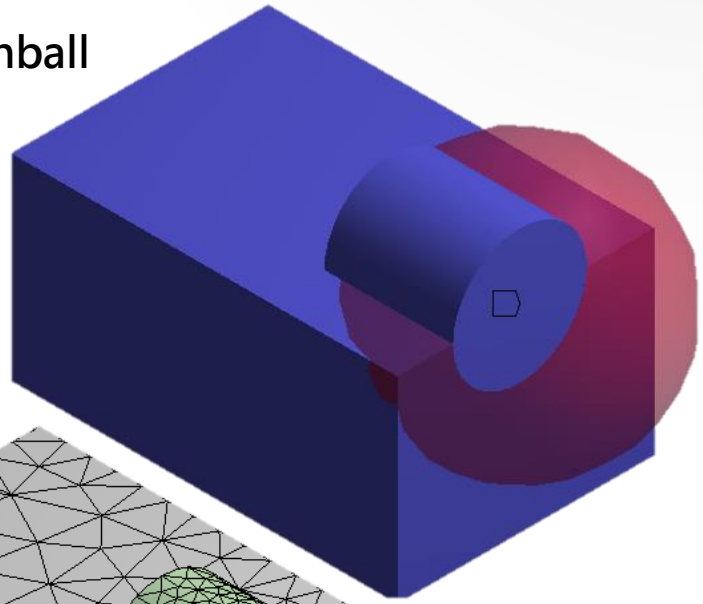




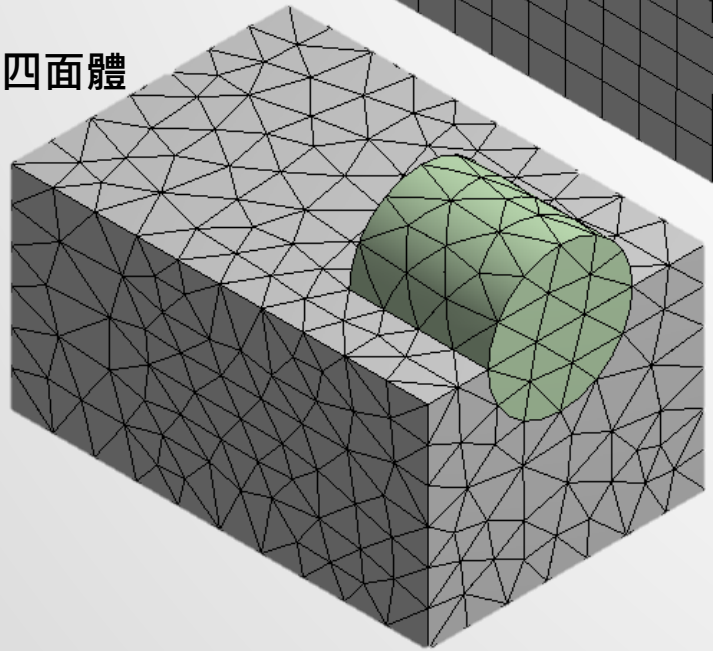
網格分割(mesh)



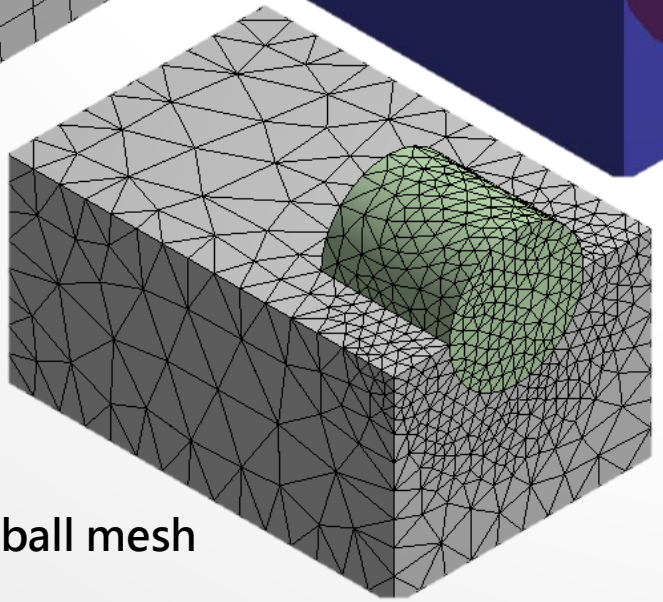
六面體



Pinball



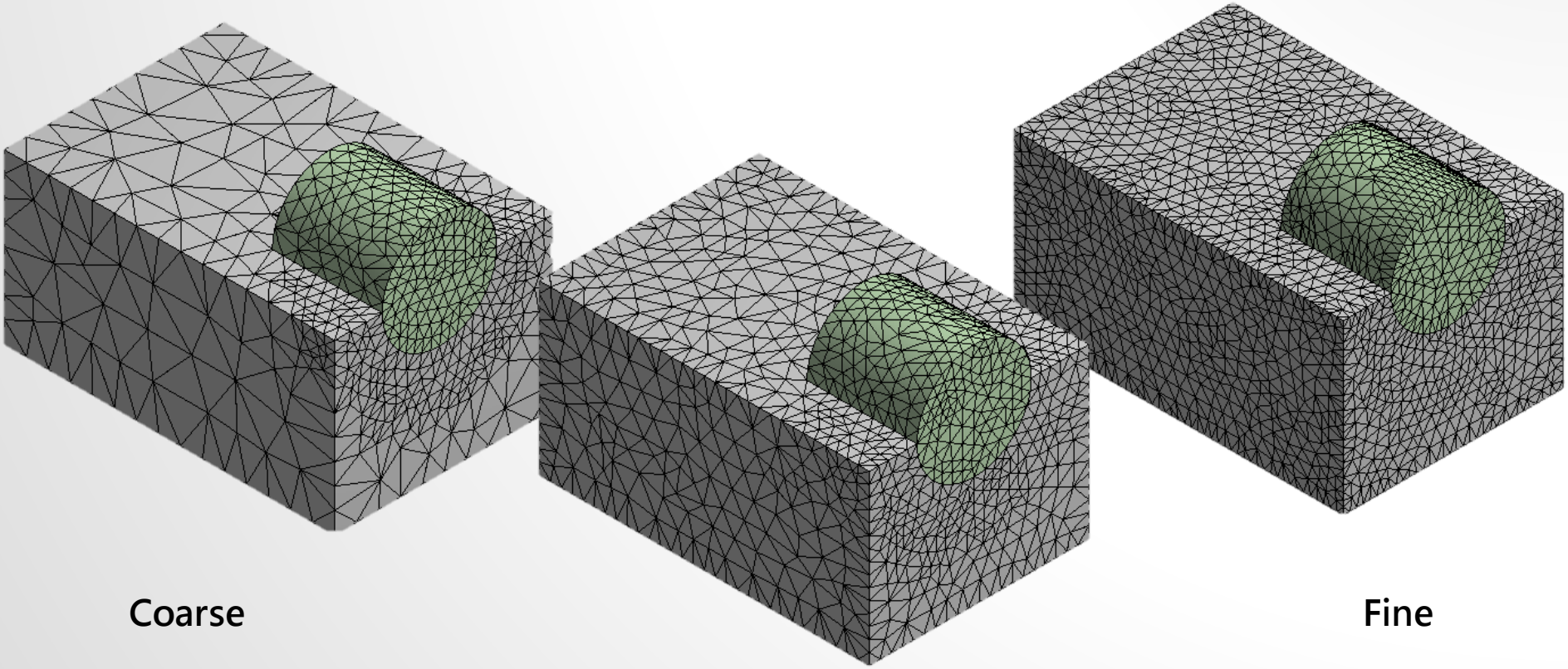
四面體



Pinball mesh

網格分割(mesh)

- Relevance Center



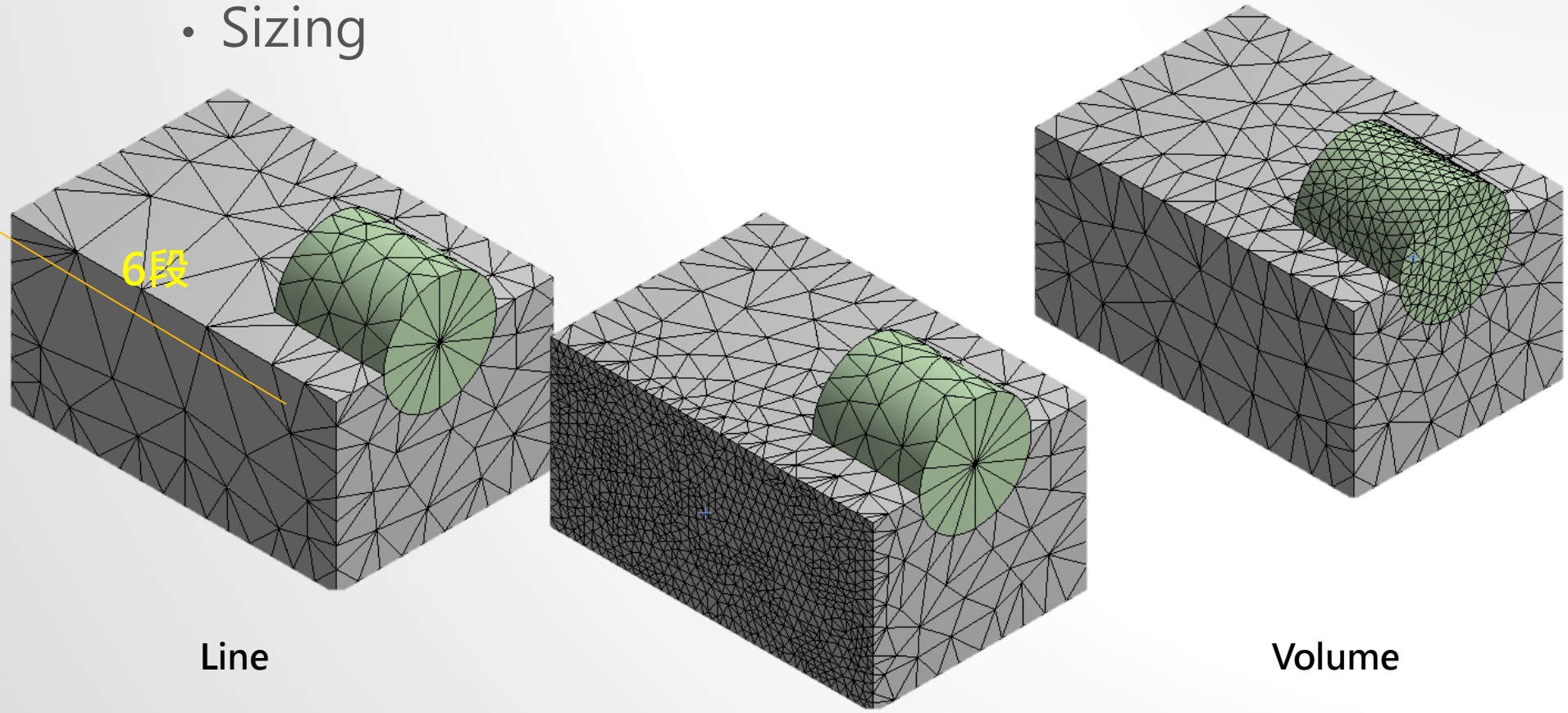
Coarse

Medium

Fine

網格分割(mesh)

- Sizing



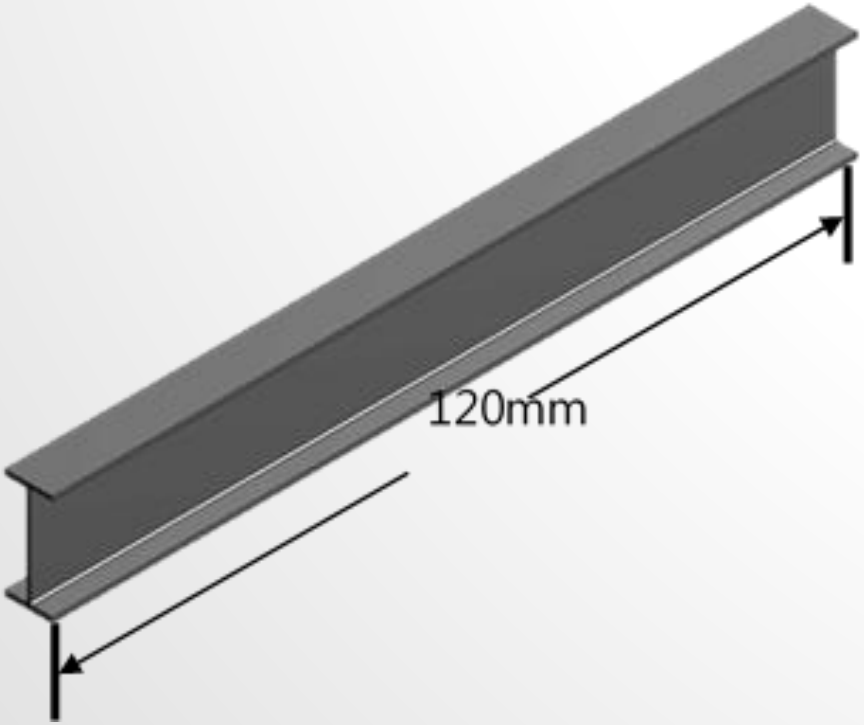
Line

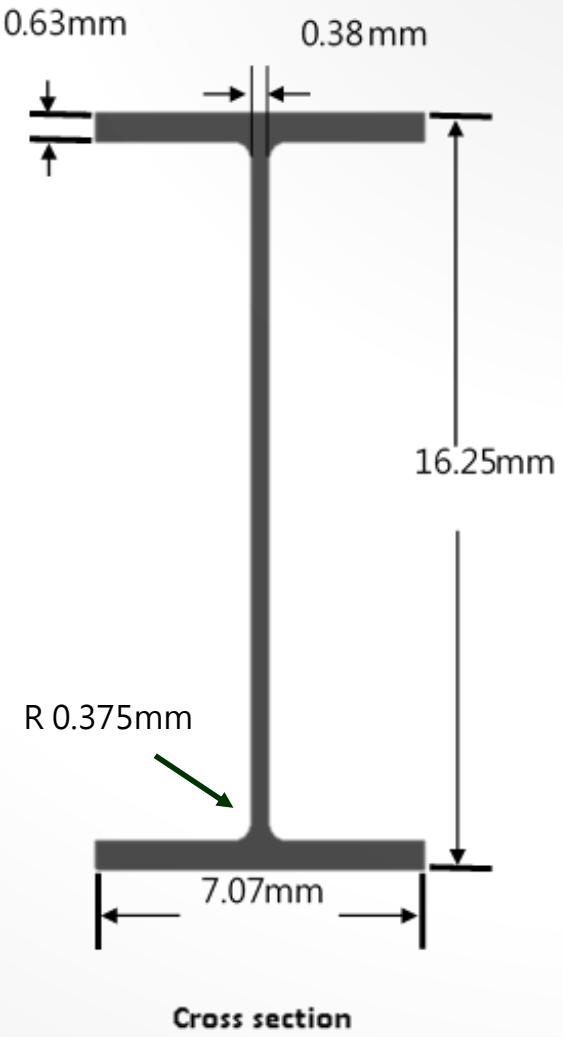
Area

Volume

Exercise 6 MESH-1 (來源：成功大學李輝煌教授)

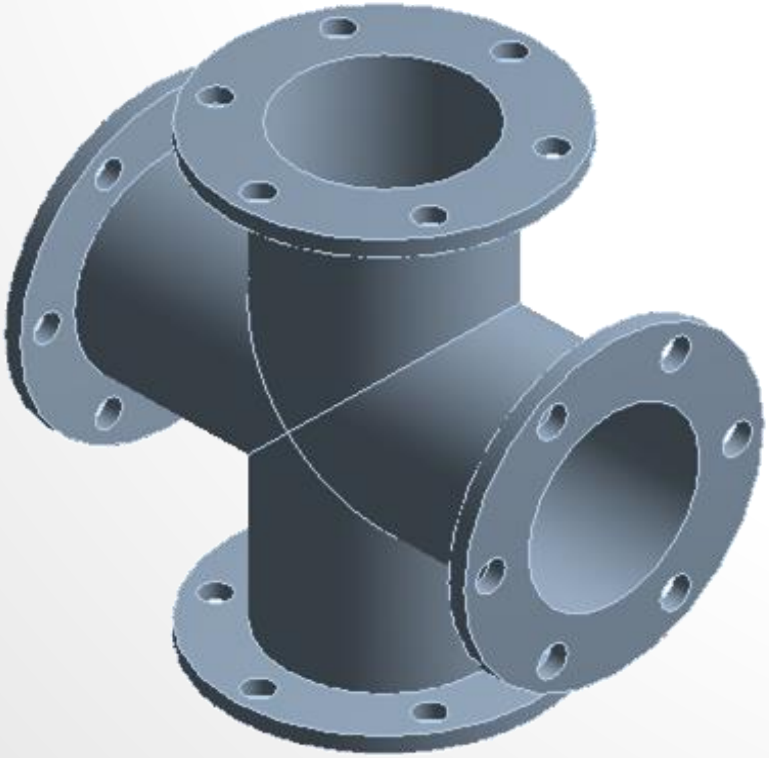
- 進行不同網格切割 **(1) 四面體(Tetrahedrons)，(2) 六面體(Hex Domain)，(3) 四面體mesh，尺寸2mm，(2)六面體，尺寸2mm**



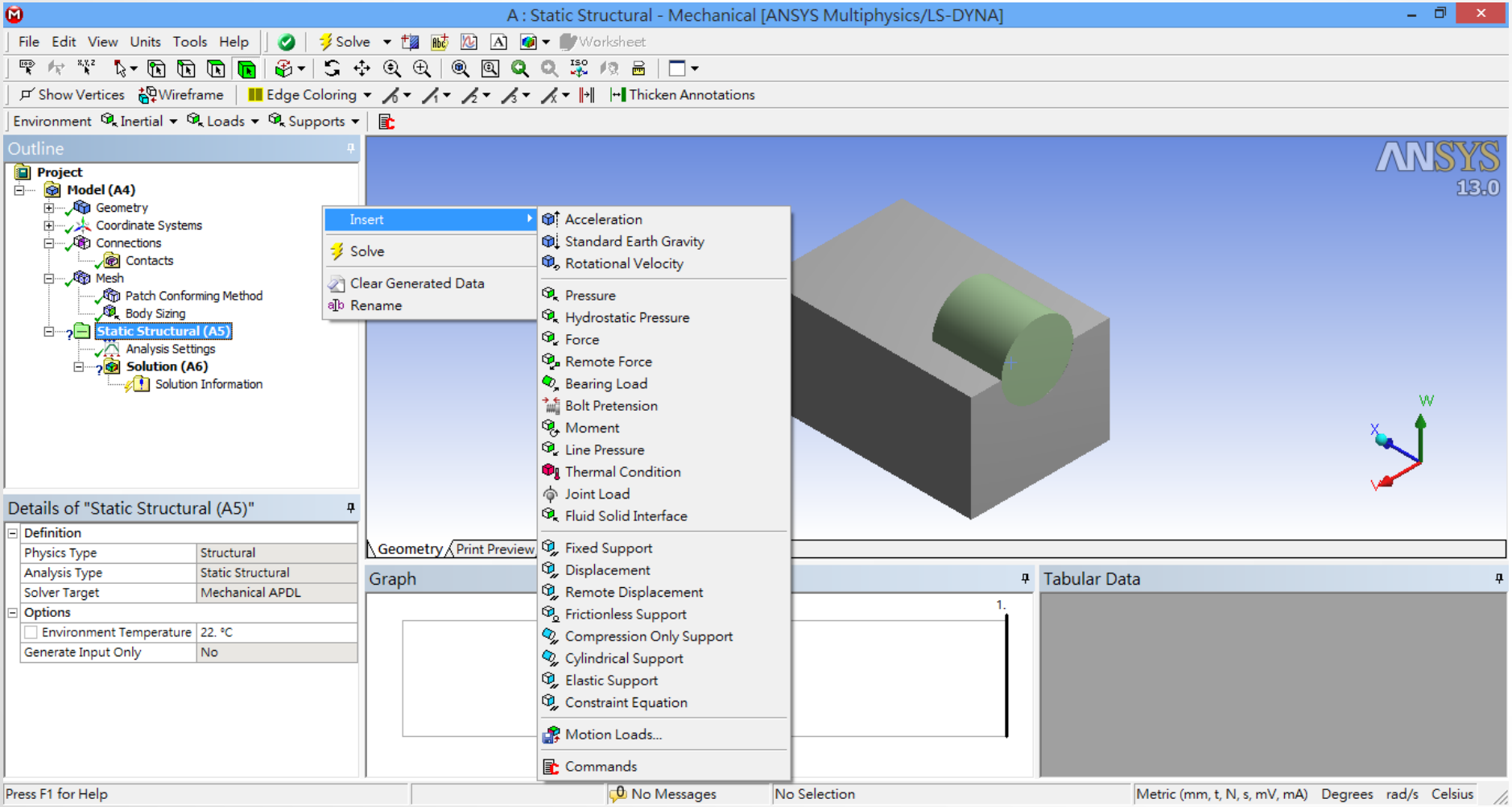


Exercise 7 MESH-2
(來源：ANSYS Workbench 有限元分析從入門到精通)

- 請依下列實體模型 (pipe.agdb) 進行不同功能之網格化練習



邊界條件給定

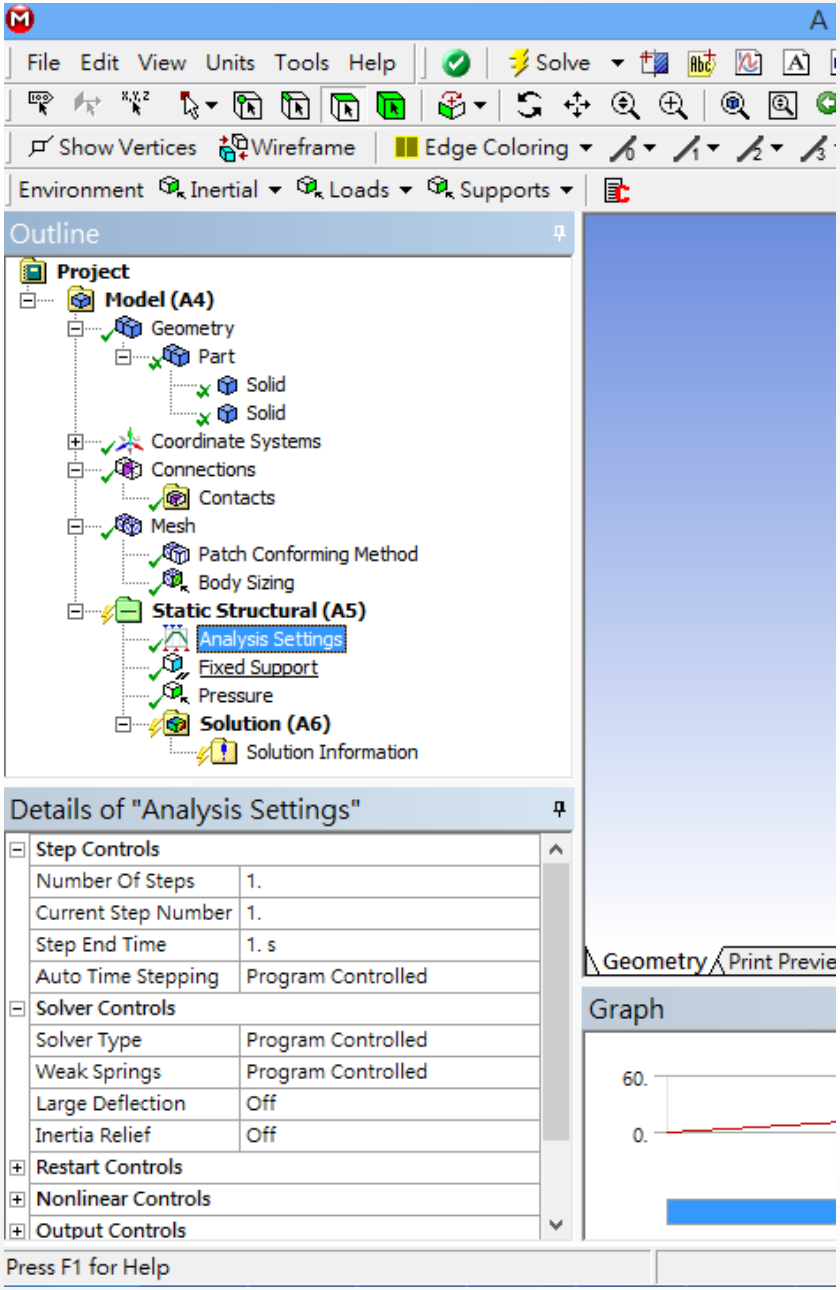

A : Static Structural - Mechanical [ANSYS Multiphysics/LS-DYNA]
File Edit View Units Tools Help
Solve
Worksheet
Show Vertices
Wireframe
Edge Coloring
Thicken Annotations
Environment
Inertial
Loads
Supports
Outline
Project
Model (A4)
Geometry
Coordinate Systems
Connections
Contacts
Mesh
Patch Conforming Method
Body Sizing
Static Structural (A5)
Analysis Settings
Solution (A6)
Solution Information
Insert
Solve
Clear Generated Data
Rename
Acceleration
Standard Earth Gravity
Rotational Velocity
Pressure
Hydrostatic Pressure
Force
Remote Force
Bearing Load
Bolt Pretension
Moment
Line Pressure
Thermal Condition
Joint Load
Fluid Solid Interface
Fixed Support
Displacement
Remote Displacement
Frictionless Support
Compression Only Support
Cylindrical Support
Elastic Support
Constraint Equation
Motion Loads...
Commands
ANSYS
13.0
Geometry
Print Preview
Graph
Tabular Data
Details of "Static Structural (A5)"
Definition
Physics Type Structural
Analysis Type Static Structural
Solver Target Mechanical APDL
Options
Environment Temperature 22. °C
Generate Input Only No
Press F1 for Help
No Messages
No Selection
Metric (mm, t, N, s, mV, mA) Degrees rad/s Celsius

解題條件設定

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

Time Control

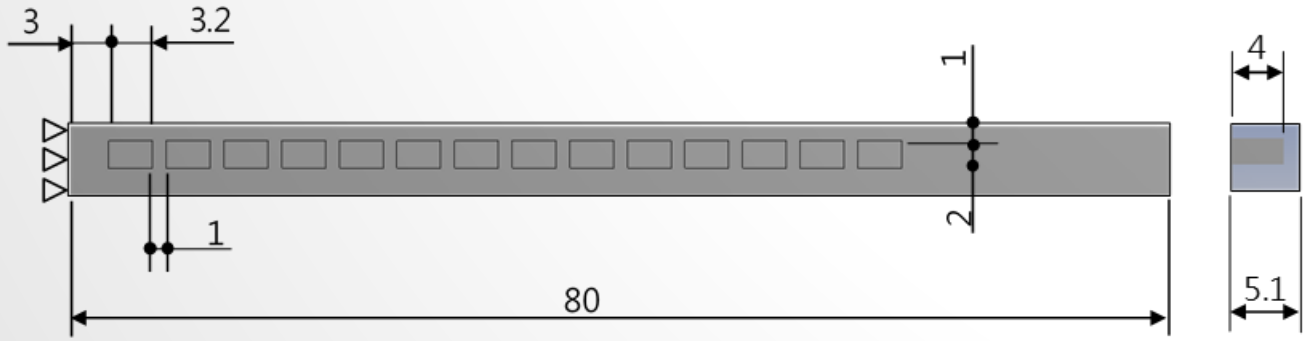
- 解題形式
- 大變形等

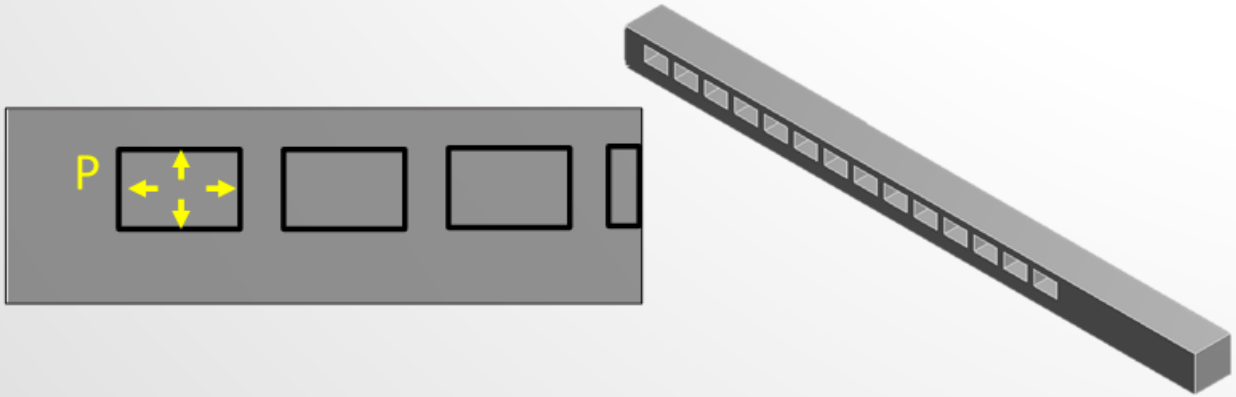


Exercise 8 MESH-3

(來源：成功大學李輝煌教授)

- Pneumatic Fingers 模型 (Symmetry) 如圖所示，請依不同網格形式進行切割 (1)自動網格(Automatic)、(2) 掃描網格(Sweep)、進行線性與非線性的分析。將pneumatic fingers 左側固定，chamber空間施以0.2MPa壓力，觀察其Y軸位移量變化。楊氏係數E=2MPa，樸松比Poisson’s ratio=0.48

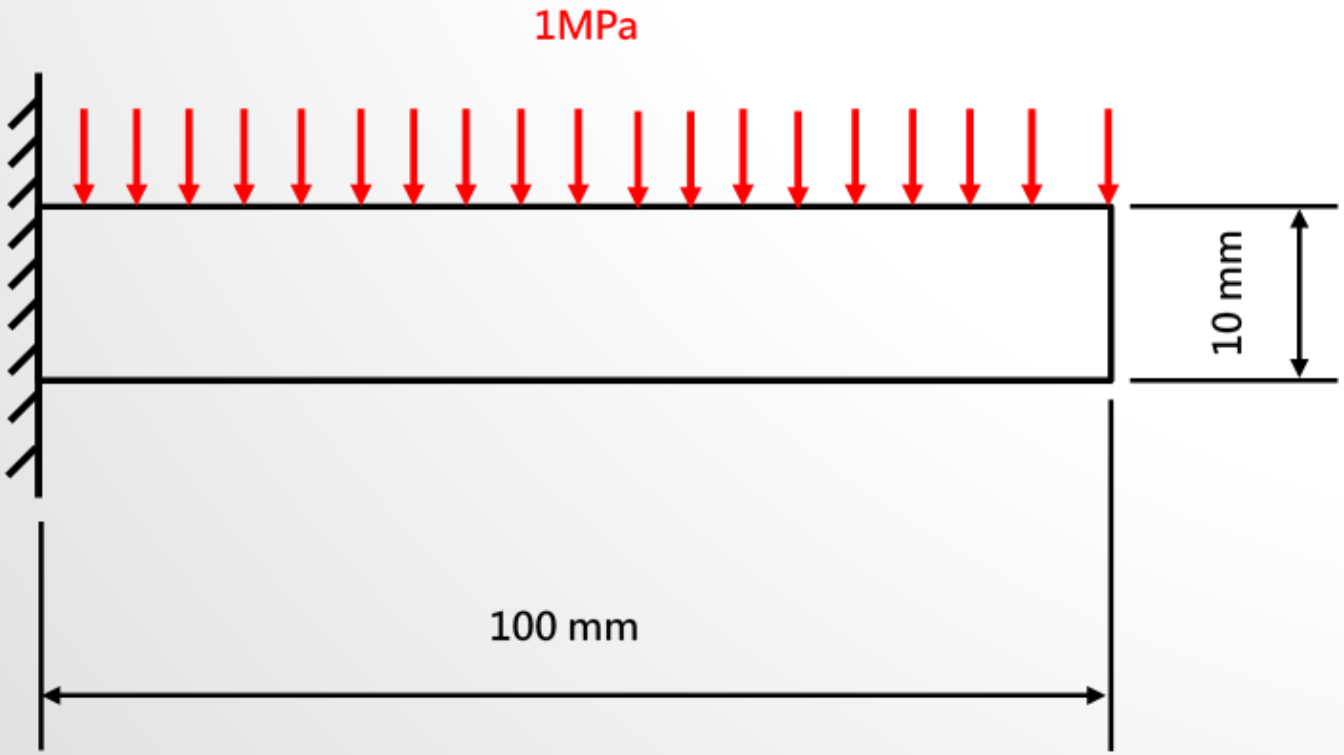




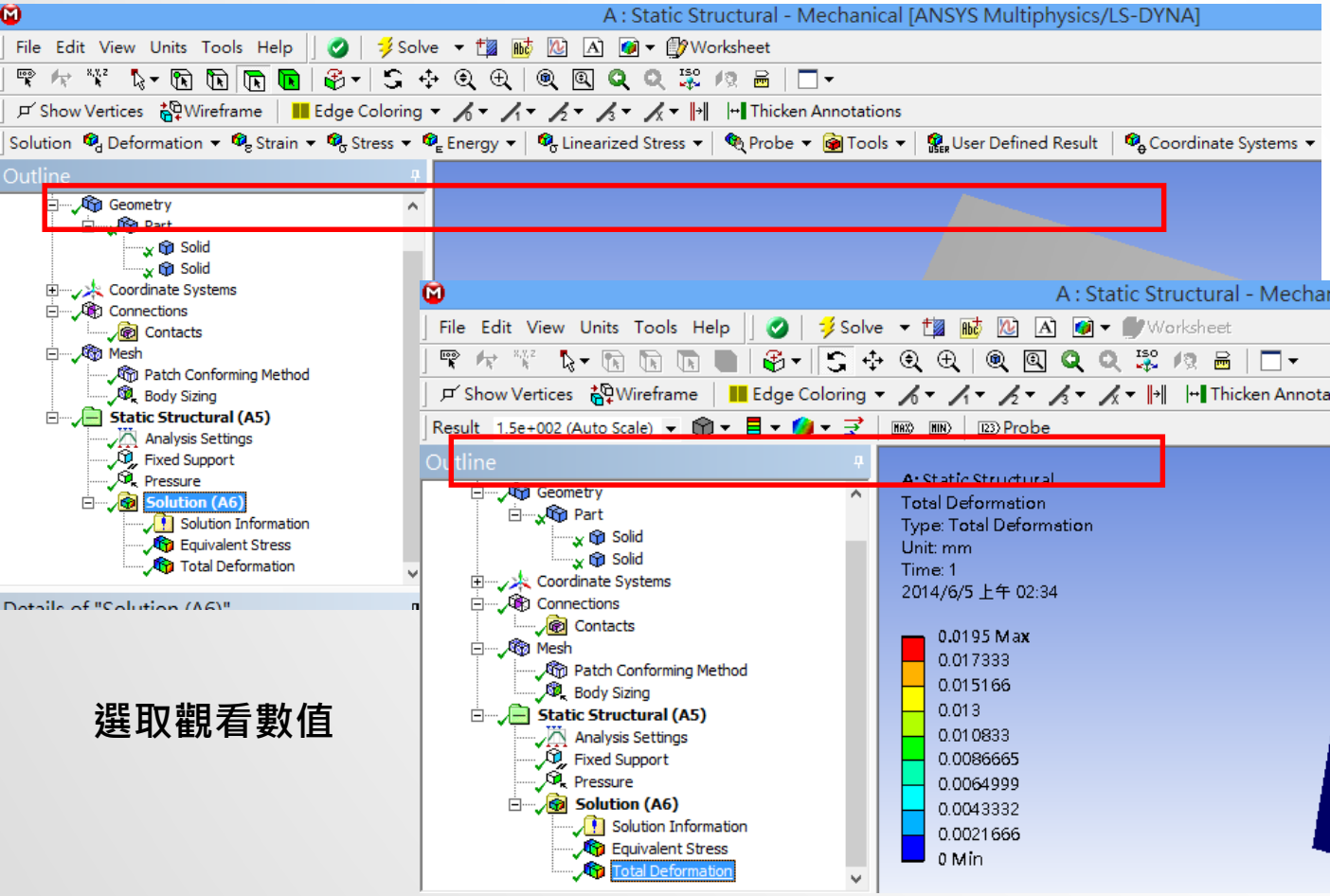
Exercise 9 MESH-4-CONVERGENCE

(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100X10X10 mm，上端平面施以1MPa均佈負載，請應用不同高/低階元素及不同元素大小(element size)探討懸臂樑模型之收斂性 (1)自動網格(Automatic)、(2) 掃描網格(Sweep)、(3)六面體(Hex Dominant)



後處理(Post-processing)

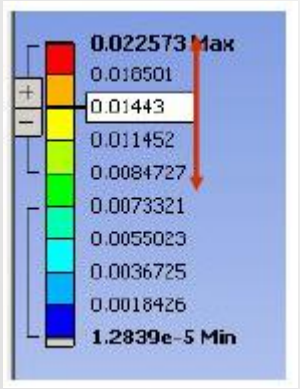


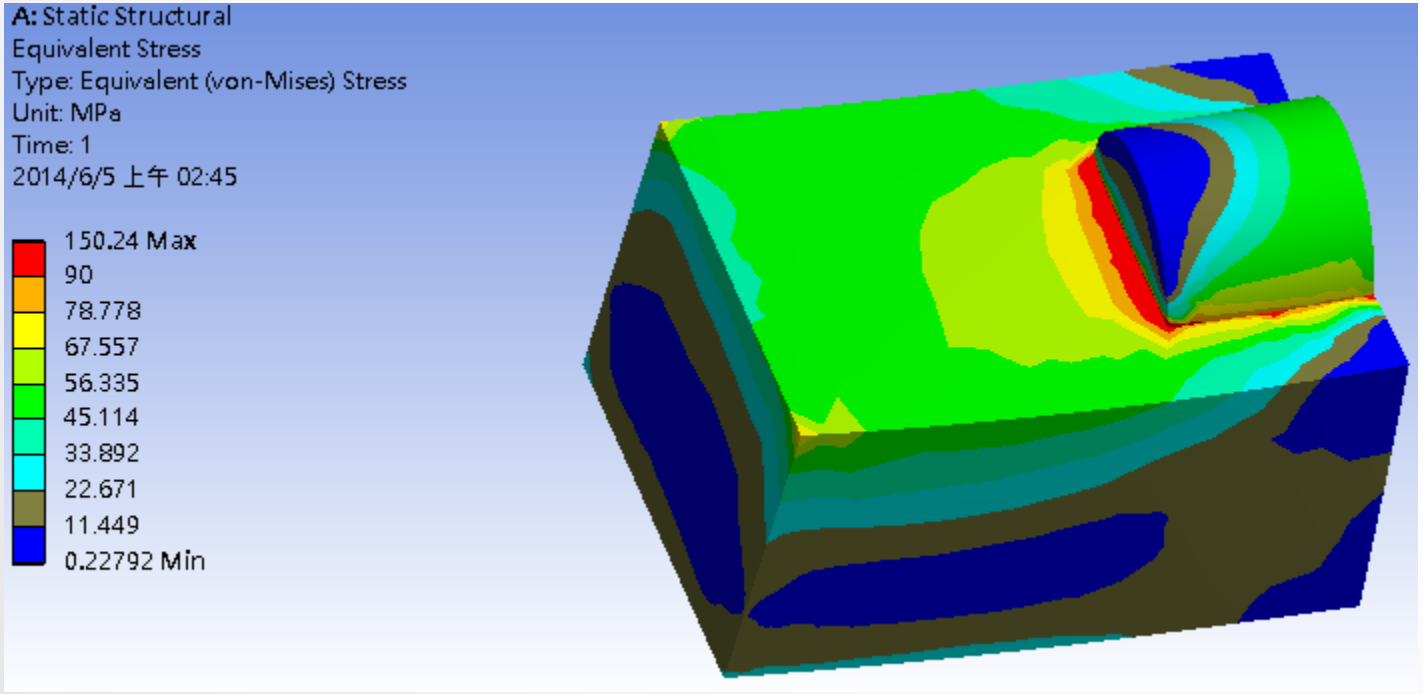
選取觀看數值

設定顯示方式

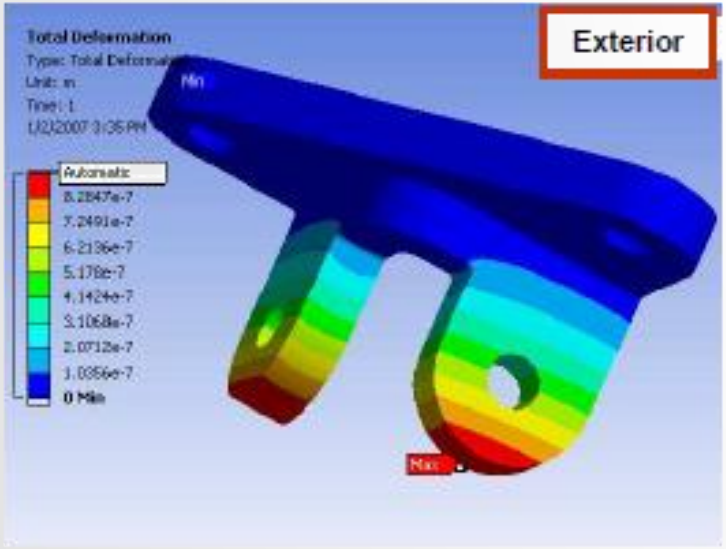
Contour 控制

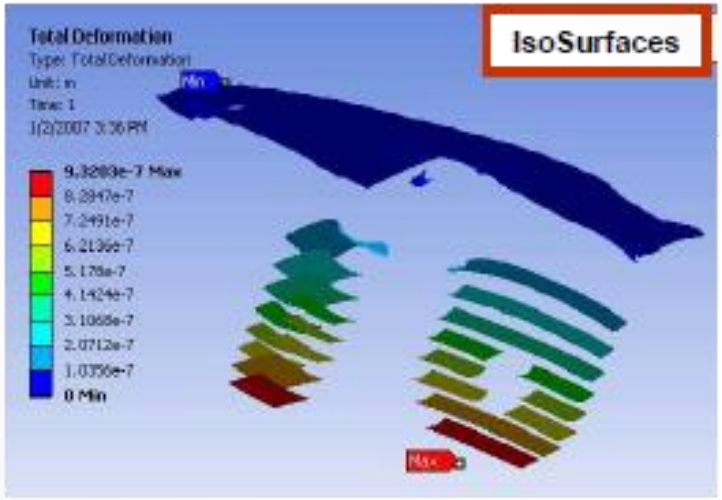
- 在圖形上之contour按右鍵可進行設定
  - 自訂數值
  - 增加色塊
  - 自訂色彩

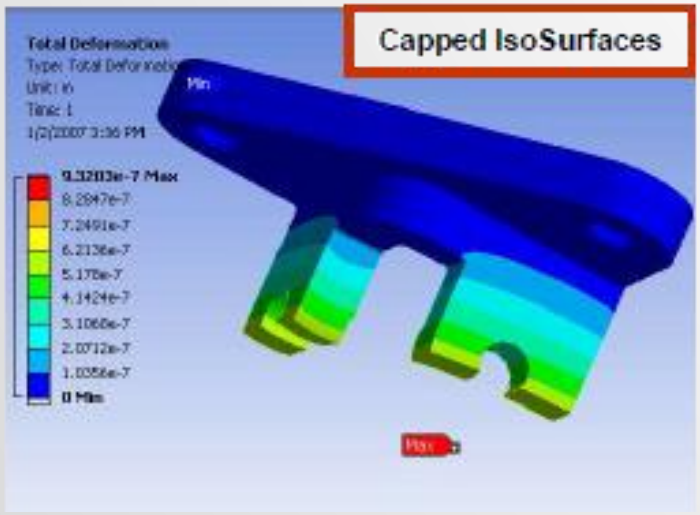


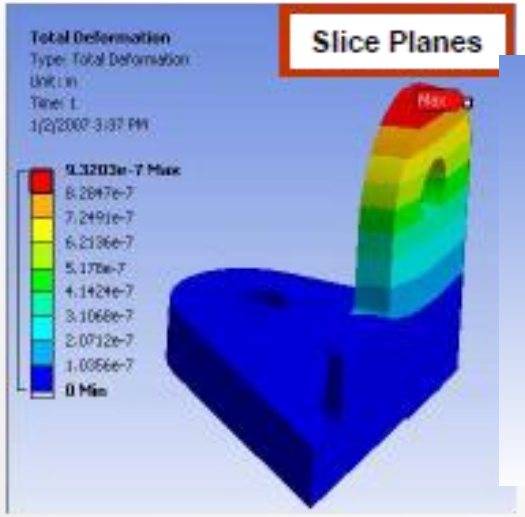


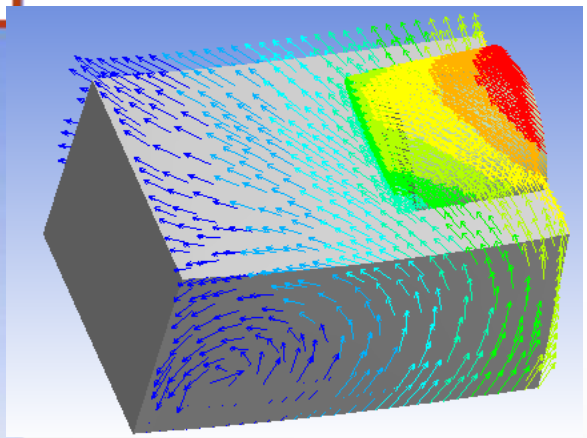
結果顯示方式


Exterior
Total Deformation
Min
Max


IsoSurfaces
Total Deformation
Min
Max

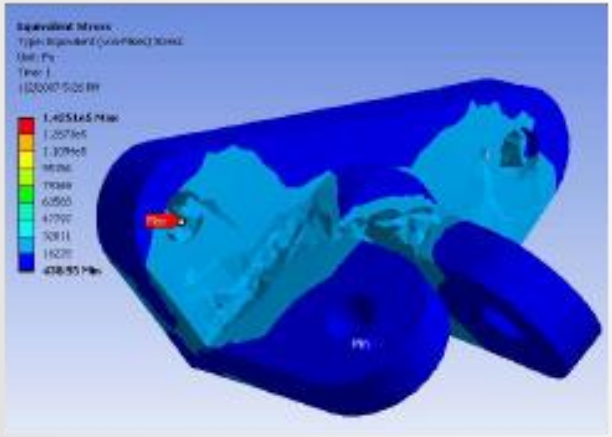

Capped IsoSurfaces
Total Deformation
Min
Max


Slice Planes
Total Deformation
Max

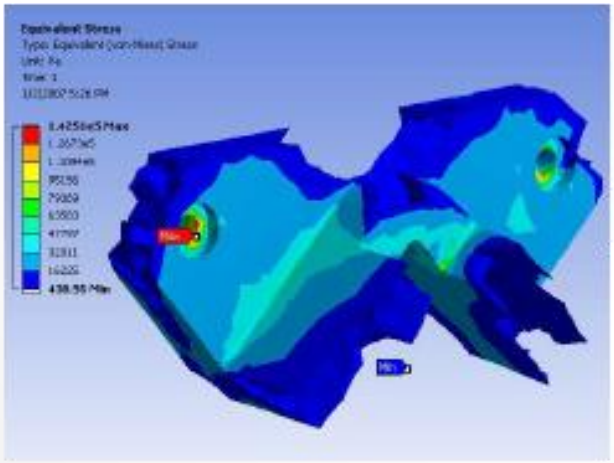


Capped ISO surface

- 可設定閥值以外區域之圖案不顯示
  - Top capped 超過閥值區域不顯示
  - Bottom capped 低於閥值區域不顯示

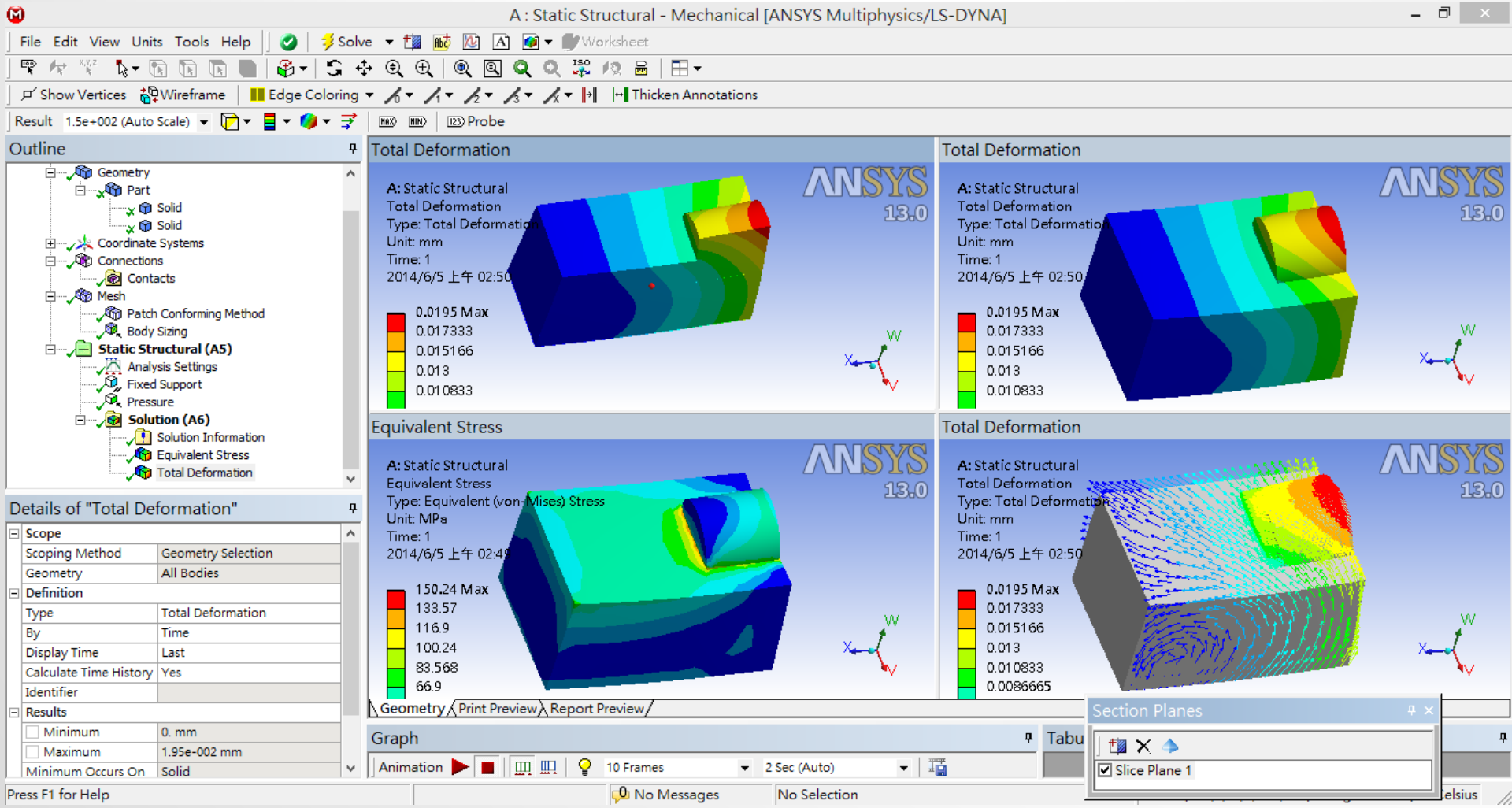


顶部封顶等值面

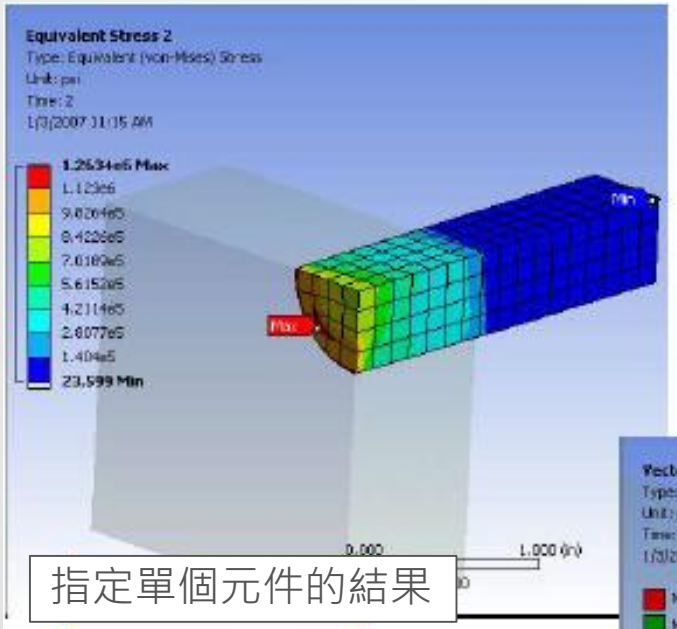


底部封顶等值面

多觀察視窗

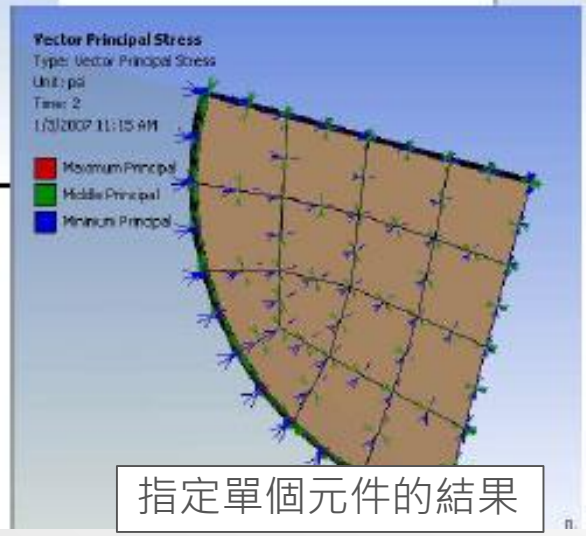

A : Static Structural - Mechanical [ANSYS Multiphysics/LS-DYNA]
File Edit View Units Tools Help
Solve
Worksheet
Show Vertices
Wireframe
Edge Coloring
Thicken Annotations
Result 1.5e+002 (Auto Scale)
Probe
Outline
Geometry
Part
Solid
Solid
Coordinate Systems
Connections
Contacts
Mesh
Patch Conforming Method
Body Sizing
Static Structural (A5)
Analysis Settings
Fixed Support
Pressure
Solution (A6)
Solution Information
Equivalent Stress
Total Deformation
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
Results
Minimum 0. mm
Maximum 1.95e-002 mm
Minimum Occurs On Solid
Total Deformation
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
2014/6/5 上午 02:50
0.0195 Max
0.017333
0.015166
0.013
0.010833
ANSYS
13.0
Total Deformation
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
2014/6/5 上午 02:50
0.0195 Max
0.017333
0.015166
0.013
0.010833
ANSYS
13.0
Equivalent Stress
A: Static Structural
Equivalent Stress
Type: Equivalent (von-Mises) Stress
Unit: MPa
Time: 1
2014/6/5 上午 02:49
150.24 Max
133.57
116.9
100.24
83.568
66.9
ANSYS
13.0
Total Deformation
A: Static Structural
Total Deformation
Type: Total Deformation
Unit: mm
Time: 1
2014/6/5 上午 02:50
0.0195 Max
0.017333
0.015166
0.013
0.010833
0.0086665
ANSYS
13.0
Geometry
Print Preview
Report Preview
Graph
Animation
10 Frames
2 Sec (Auto)
Tabu
Section Planes
Slice Plane 1
Press F1 for Help
No Messages
No Selection

指定元件觀察結果



指定單個元件的結果

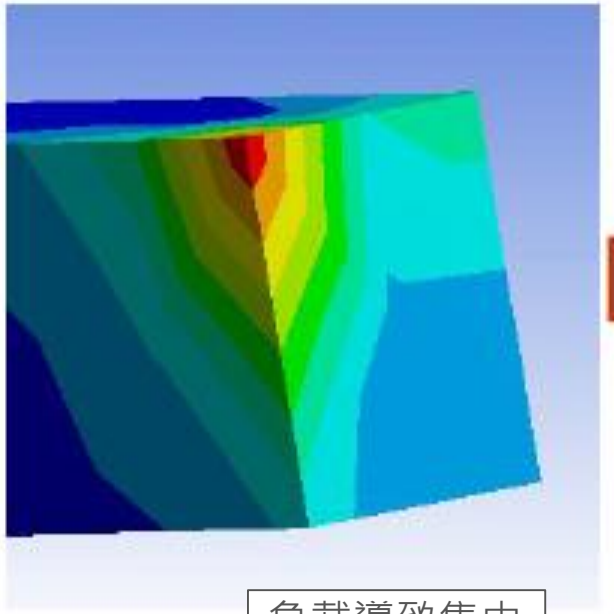
指定單個元件的結果



指定單個元件的結果

應力奇異點

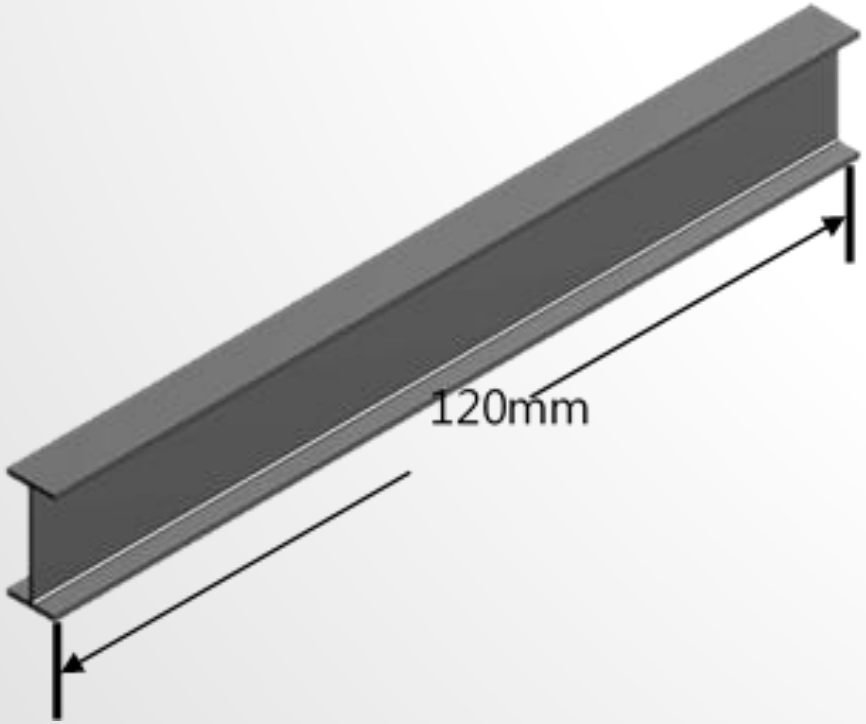
- 當結果顯示發生奇異點時，例如力量及壓力施加在模型點或線上
- 補救措施
  1. 使用New Section Plane將應力奇異點刪除
  2. 若為相反方向需點擊切開軸
  3. 於contour點右鍵，並點選進行Adjust to Visible

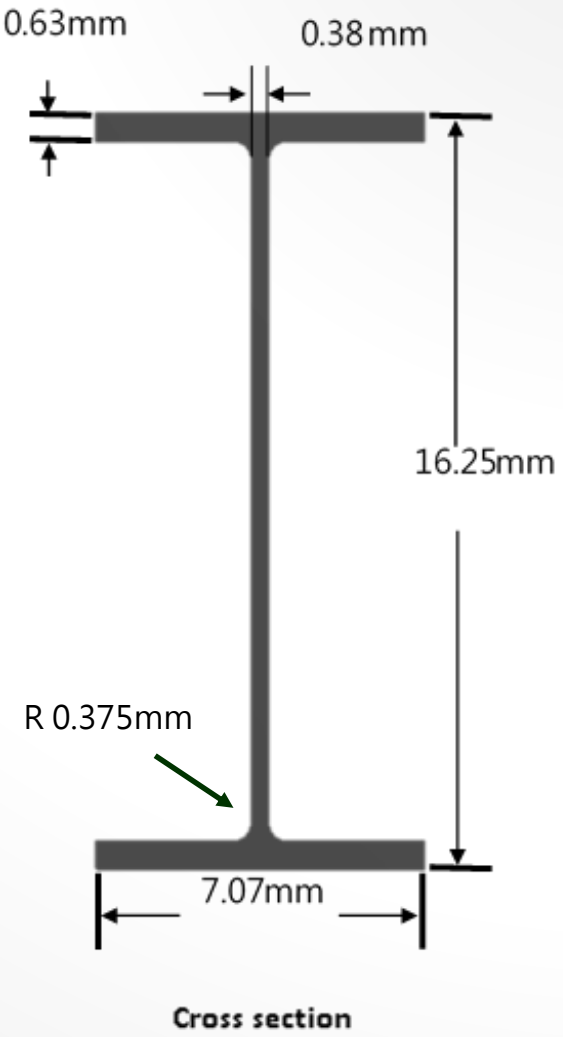


負載導致集中

Exercise 10 CAE-3 (來源：成功大學李輝煌教授)

- 於樑一端進行固定，另一端施予一壓力100MPa。觀察其等效應力及位移量變化。材料特性：楊氏係數：110000 MPa，浦松比：0.3

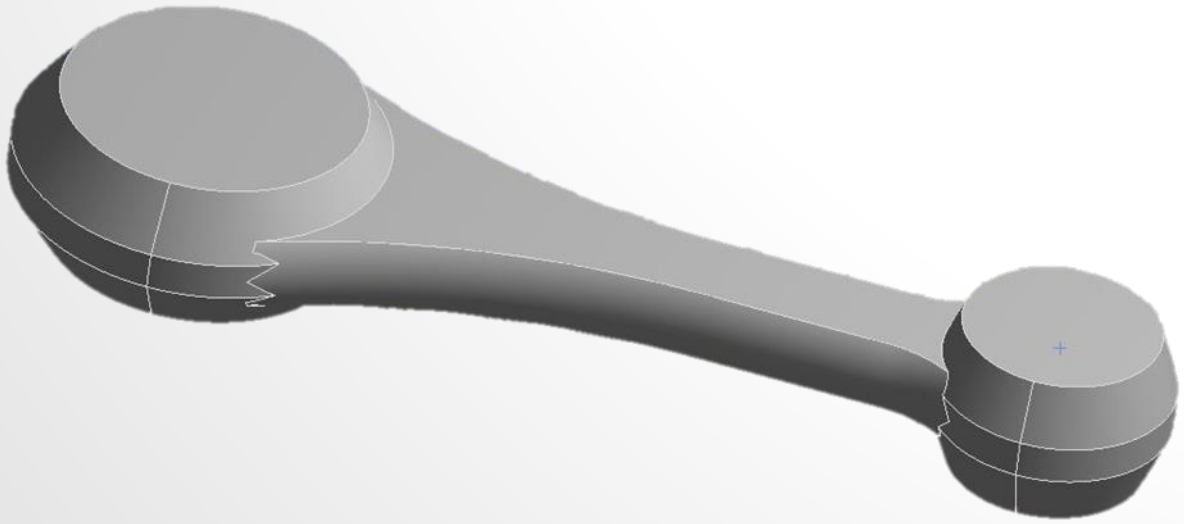




Exercise 11 CAE- 4

(來源：ANSYS Workbench 有限元分析從入門到精通)

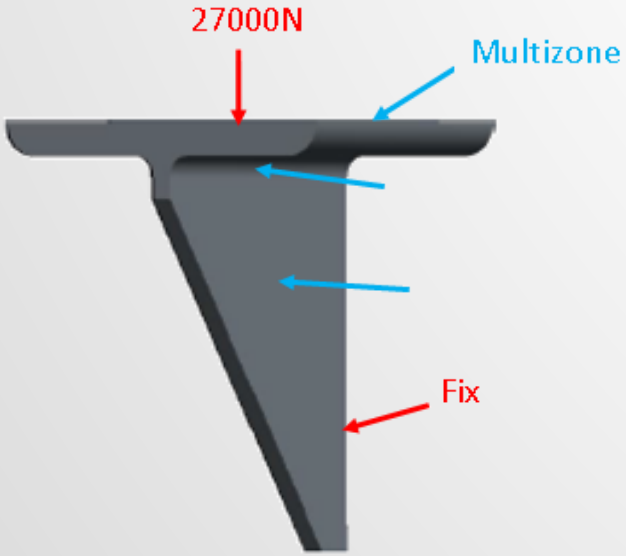
連桿基體模型由外部導入，ELEMENT SIZE為10，於兩大圓面積進行固定，另一端施予一外力1000N。觀察其等效應力及位移量變化。材料特性：灰鑄鐵(GRAY CAST IRON)

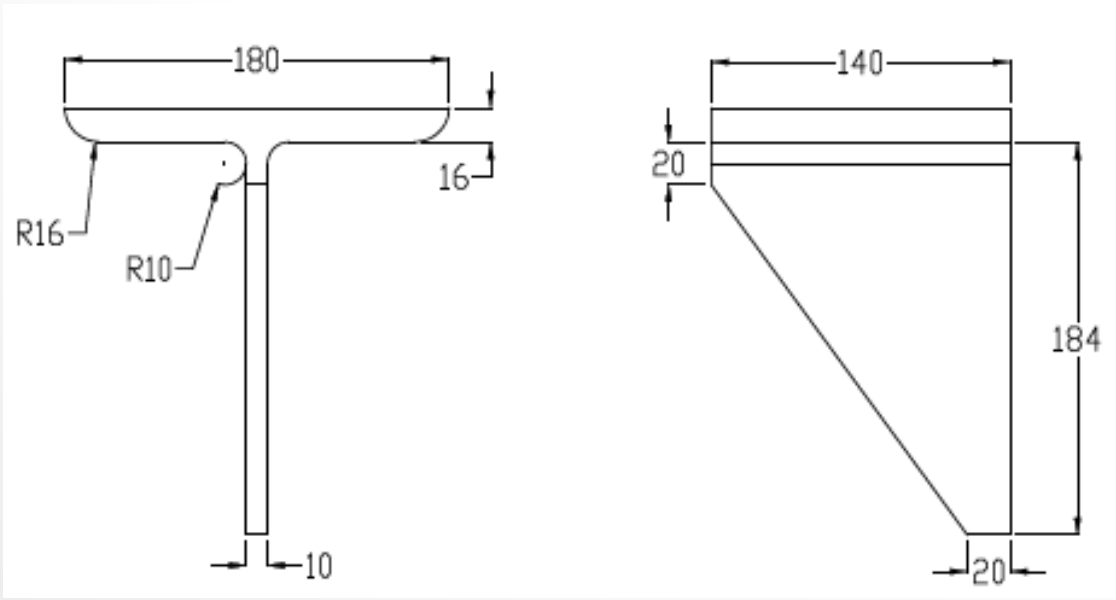


Exercise 12 CAE- 5-機尾

(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)

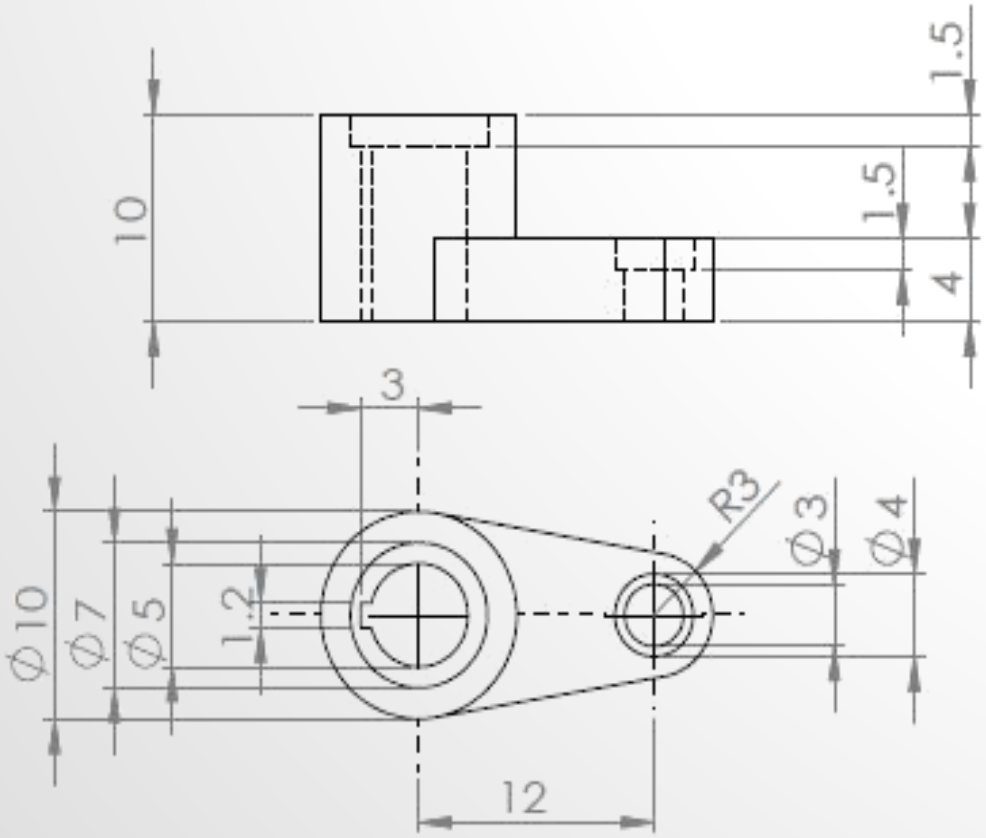


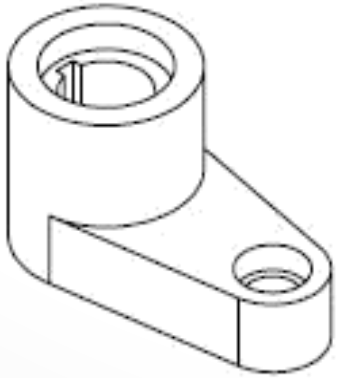


Exercise 13 CAE- 6

(來源：ANSYS Workbench 有限元分析從入門到精通)

聯軸器模型，尺寸如圖所示，聯軸器在底面的四周邊界不能發生上下運動；在底面的兩個圓周尚不能發生任何方向的運動；在大軸孔的鍵槽的一側受到1E5PA的壓力。觀察其等效應力及位移量變化。

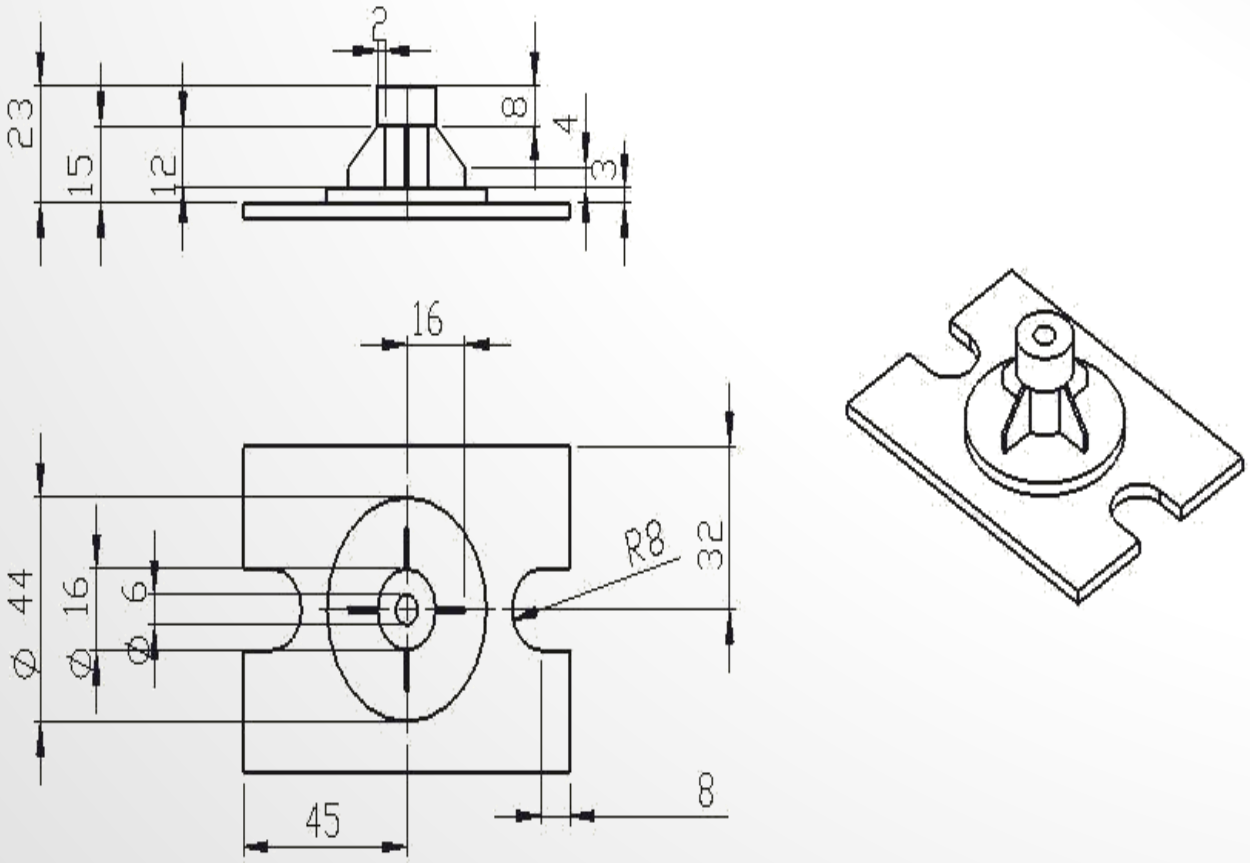




Exercise 14 CAE- 7

(來源：ANSYS Workbench 有限元分析從入門到精通)

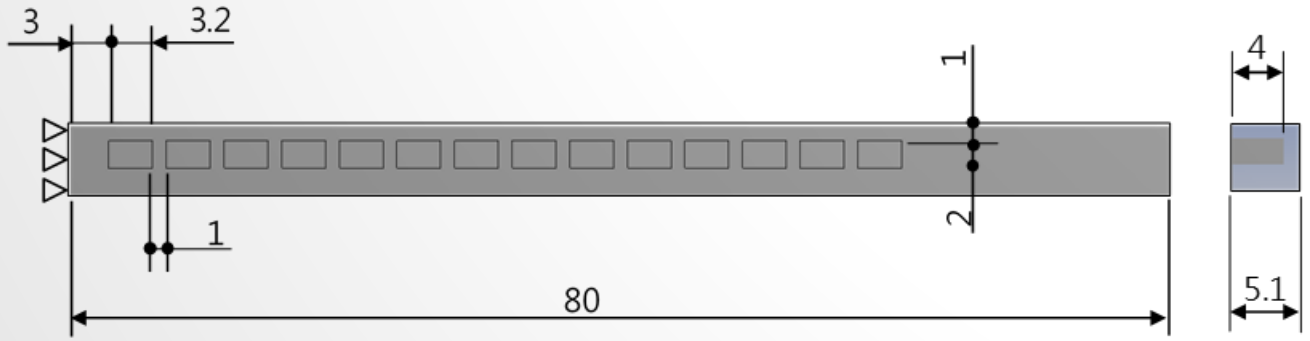
試建構機蓋模型，尺寸如圖所示並於中央孔內給一均勻往外推之250MPA壓力，觀察其等效應力、位移量變化，材料選用鋼。

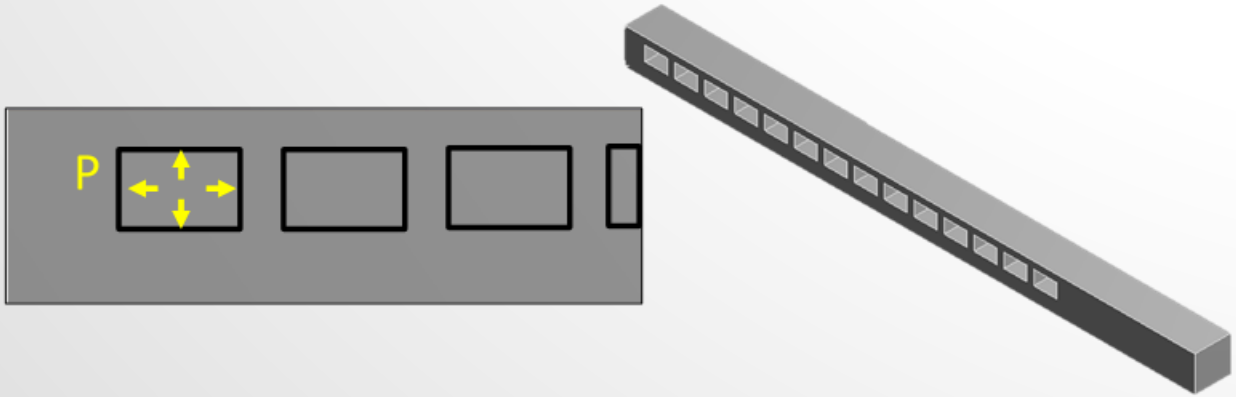


Exercise 8-1 MESH-3

(來源：成功大學李輝煌教授)

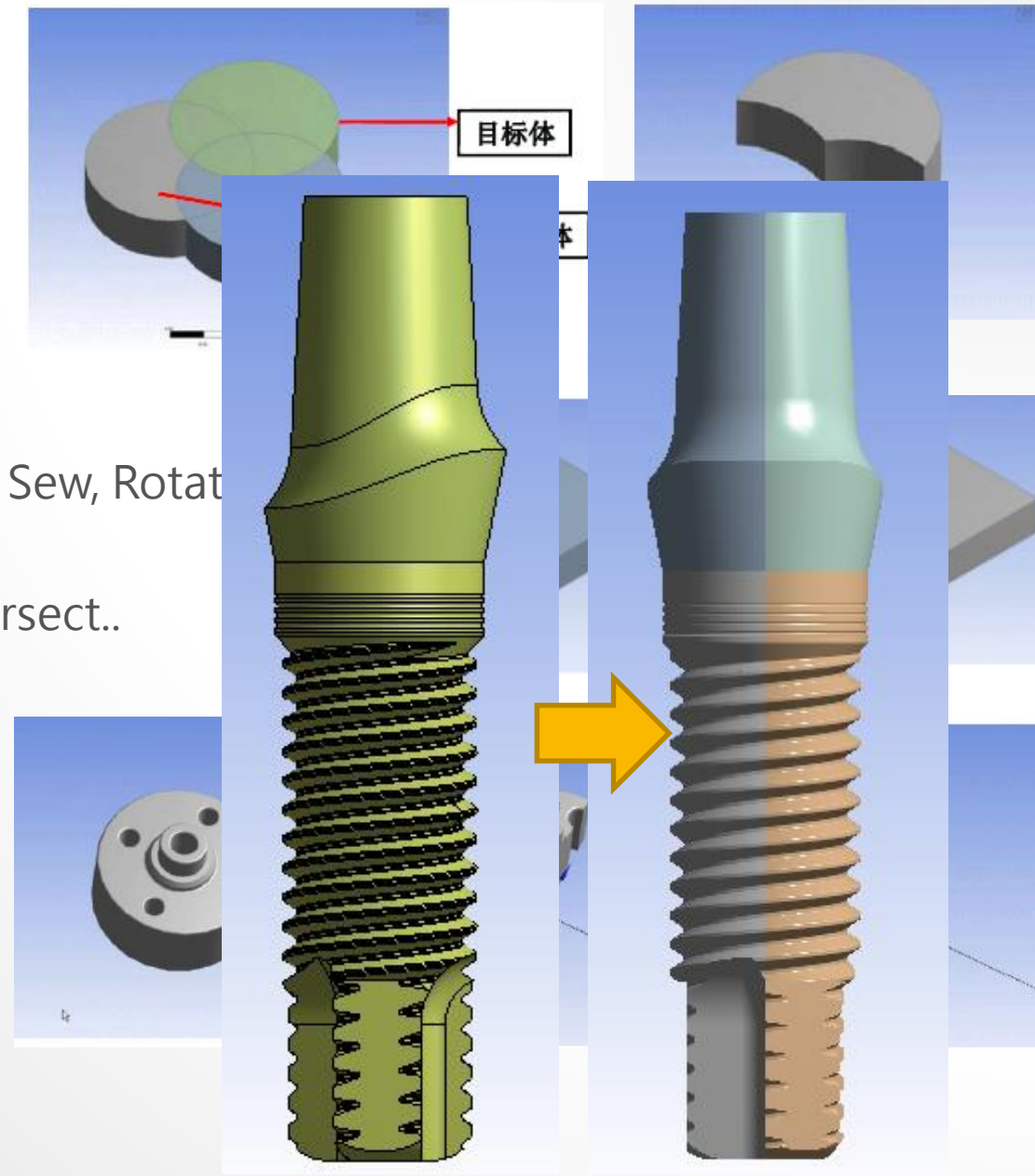
- Pneumatic Fingers 模型 (Symmetry) 如圖所示，請依不同網格形式進行切割 (1)自動網格(Automatic)、(2) 掃描網格(Sweep)並進行線性與非線性分析比較。將pneumatic fingers 左側固定，chamber空間施以0.2MPa壓力，觀察其Y軸位移量變化。楊氏係數E=2MPa，樸松比Poisson’s ratio=0.48





體積建模工具

- Create
  - Pattern
  - Body Operation
    - Mirror, Move, Scale, Sew, Rotat
  - Boolean
    - Unite, Subtract, Intersect..
  - Slice
    - By plane, By face, …
- Tool
  - Symmetry
  - Merge
  - …

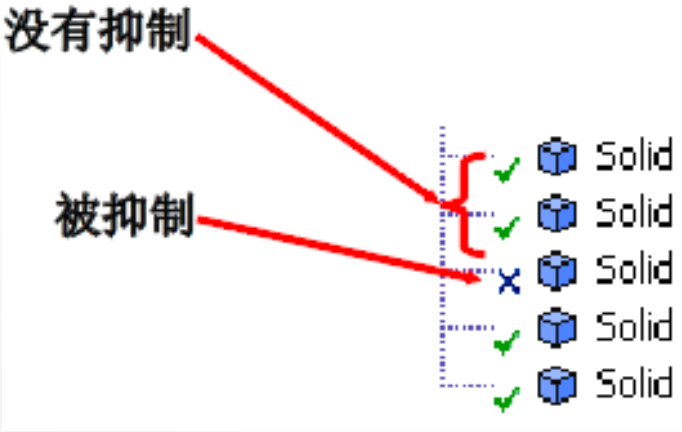


Slice

體積運算功能亦比ANSYS佳

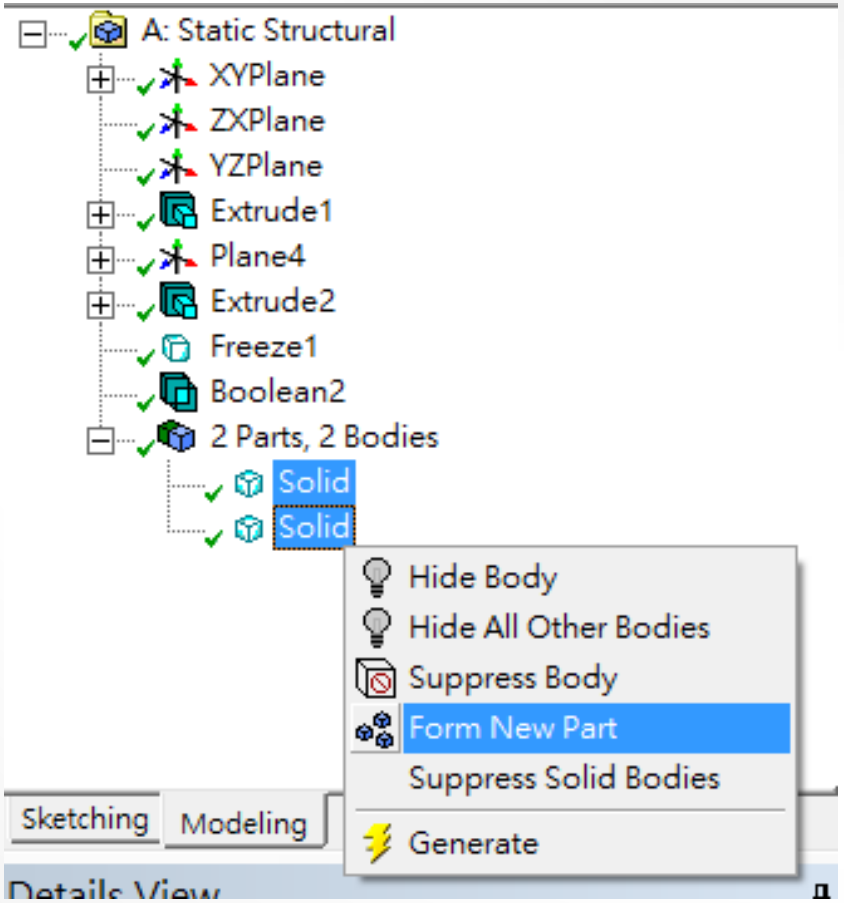
物件抑制（Suppress）

- 抑制物件不會於繪圖視窗中
- 抑制物件不會送到其他Workbench模組中用於網格分割及分析
- 抑制物件在結構樹狀視窗中前面有一个 “X”



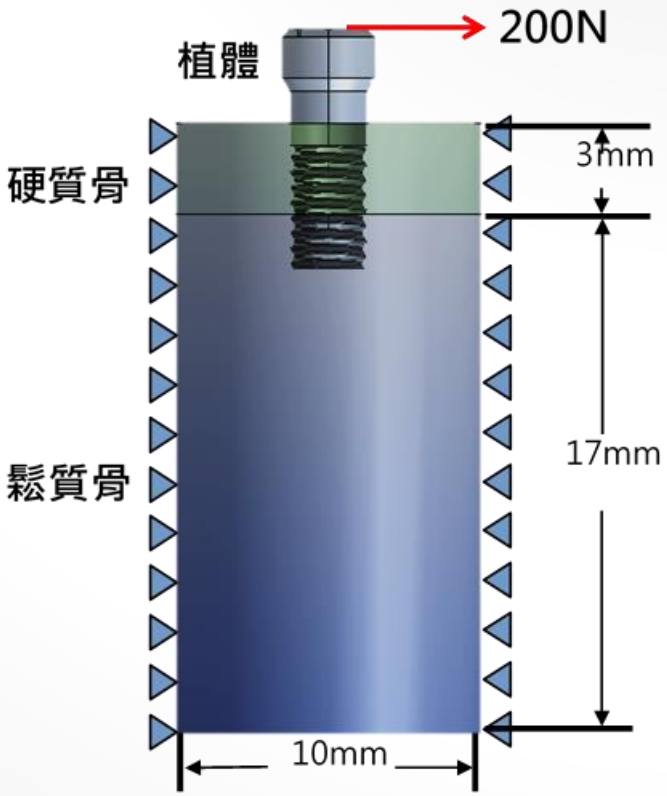
界面條件設定

- 模型為組合件時，若要組合件間能共用面，達到力量直接傳遞時，必須將此部份組件形成一個群組>From New part。
- 若要組合件間能有各自的面，達到contact效果時，則不須進行此動作，模型匯入Design Simulation時軟體會自動判斷出非連續面之部份。



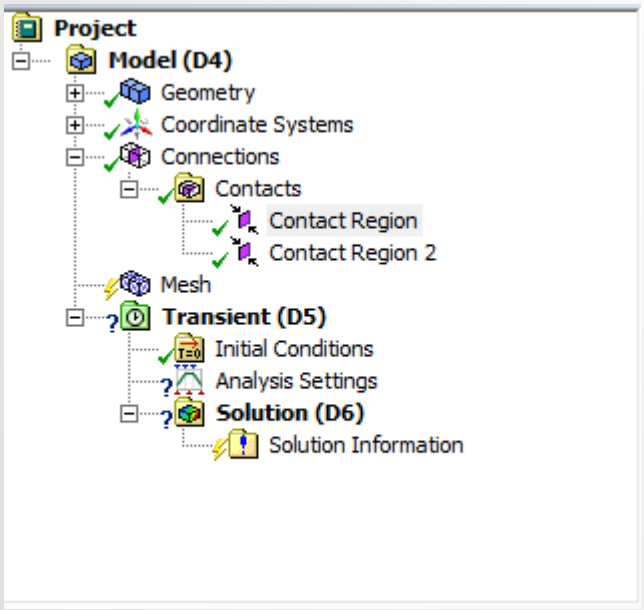
Exercise 15 CAE- 8

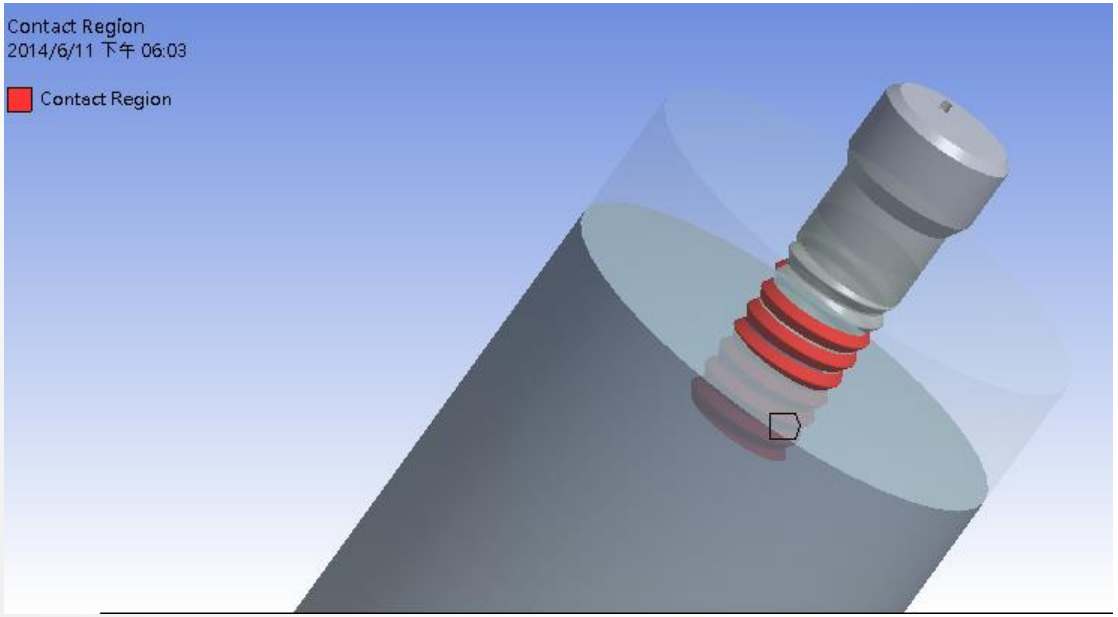
- 請構出硬質骨與鬆質骨圓柱模型，尺寸如圖所示，並將外部CAD軟體建構出之植體檔(screw.iges)匯入，各材料特性硬質骨(楊氏係數=17000 MPa；蒲松比=0.3)、鬆質骨(楊氏係數=200 MPa；蒲松比=0.2)及植體(鈦合金楊氏係數=110000 MPa；蒲松比=0.33)，並施加側向力200N負載於植體頂部(已於植體頂部建構一凹點特徵)上，並設定硬質骨/鬆質骨外側自由度為0 (如下圖)。
  - (1) 請將模型進行兩種網格分割(mesh)(包含粗糙網格(網格尺寸植體=0.5mm；硬質骨=0.8mm、鬆質骨=1.0mm)及精緻網格(網格尺寸植體=0.3mm；硬質骨=0.5mm、鬆質骨=0.5mm))
  - (2) 請完成植體與硬質骨/鬆質骨界面未結合(unbonded)狀態之設定(模擬植體剛植入骨頭)，(3)請完成植體與硬質骨/鬆質骨界面結合(bonded)狀態之設定(模擬植體與骨頭已骨整合)，並觀察其狀態下之硬質骨最大主應變(Maximum Principal strain)及植體最大等效應力(von-Mises stress)情形。



接觸(contact)非線性分析

- 於Design Simulation會偵測到非Bonded之界面，並於Connection中顯示所有之contact區域





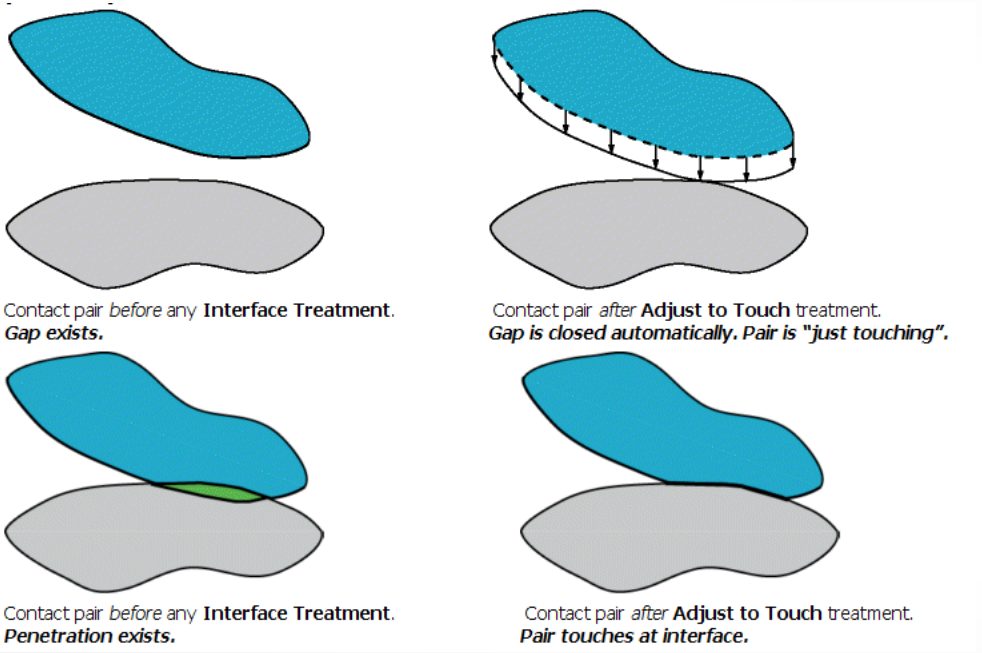
接觸(contact)非線性分析

- 接觸種類設定
  - Bonded
    - 預設項目，沒有相對滑動和分離，會忽略初始穿刺(penetration)，模擬為相互連接
  - No Separation
    - 此設定類似Bonded，僅適用於3D（面）或2D（邊）之接觸，沒有相對分離，僅可延接觸面有些微無摩擦滑動
  - Frictionless
    - 此為單邊接觸，假設摩擦係數為0，允許相對滑動，出現分離時法向量壓力為0，法向會分離
  - Rough
    - 此設定類似frictionless，有摩擦係數，無相對滑動，法向會分離
  - Frictional
    - 有摩擦係數，有相對滑動，法向會分離

接觸(contact)非線性分析

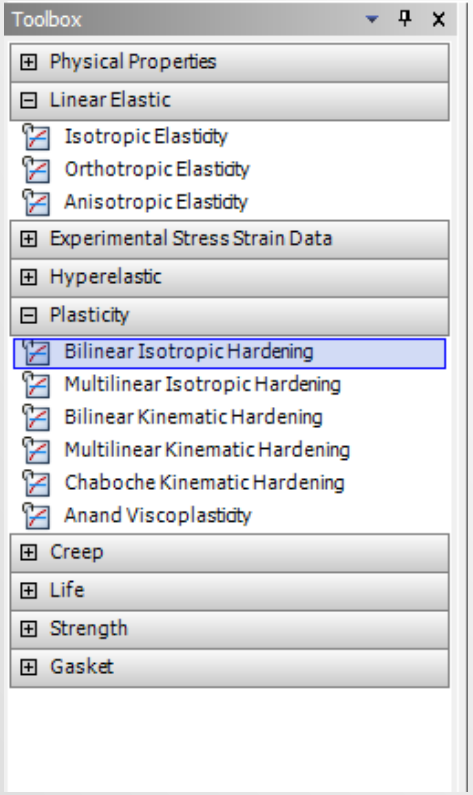
- Interface treatment
  - **Offset**：给初始調整給定一个**0**或非**0**的值
  - **Adjusted to Touch**：**ANSYS**把間隔調整到恰好接觸的位置

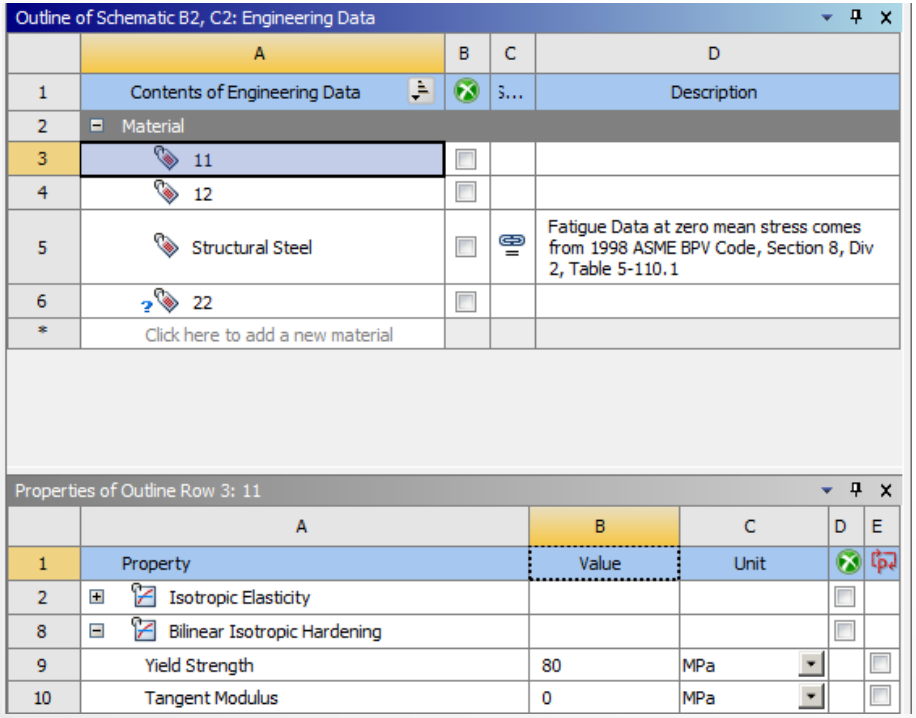
Contact	16 Faces
Target	16 Faces
Contact Bodies	Solid
Target Bodies	Solid
Definition	
Type	Frictional
Friction Coefficient	0.2
Scope Mode	Automatic
Behavior	Symmetric
Suppressed	No
Advanced	
Formulation	Pure Penalty
Interface Treatment	Adjust to Touch
Normal Stiffness	Program Controlled



非線性材料模擬分析

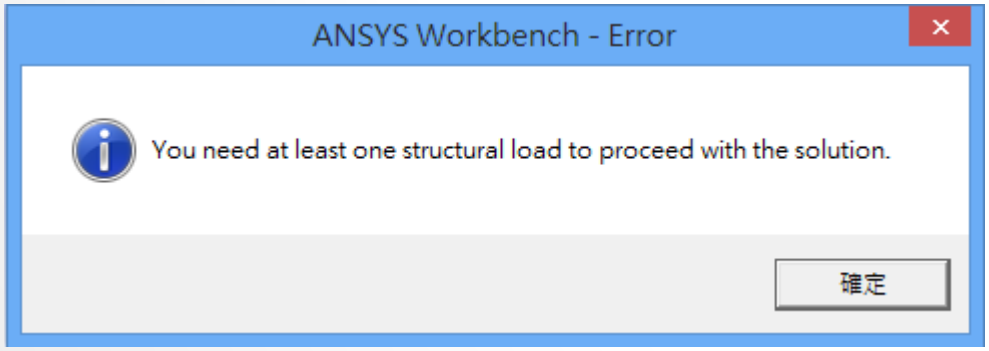
- 殘留應力(residual stress)
  - 非線性材料設定：
    - Toolbox>Plasticity>Bilinear Isotropic Hardening





非線性材料模擬分析

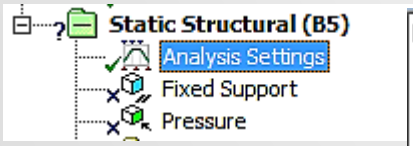
- 求解方法：
  - Workbench無法在無邊界及負載條件下進行解題



  - 利用不同Step-時間點(Time)來給定負載

Tabular Data

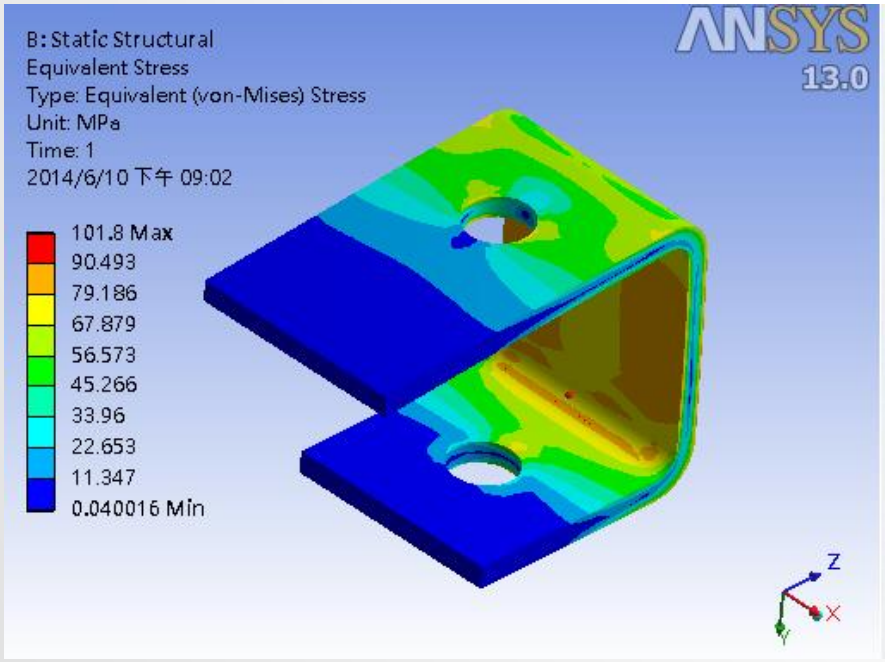
	Steps	Time [s]	X [MPa]	Y [MPa]	Z [MPa]
1	1	0.	0.	0.	0.
2	1	1.	0.	0.1	0.
3	2	2.	0.	0.	0.
*					



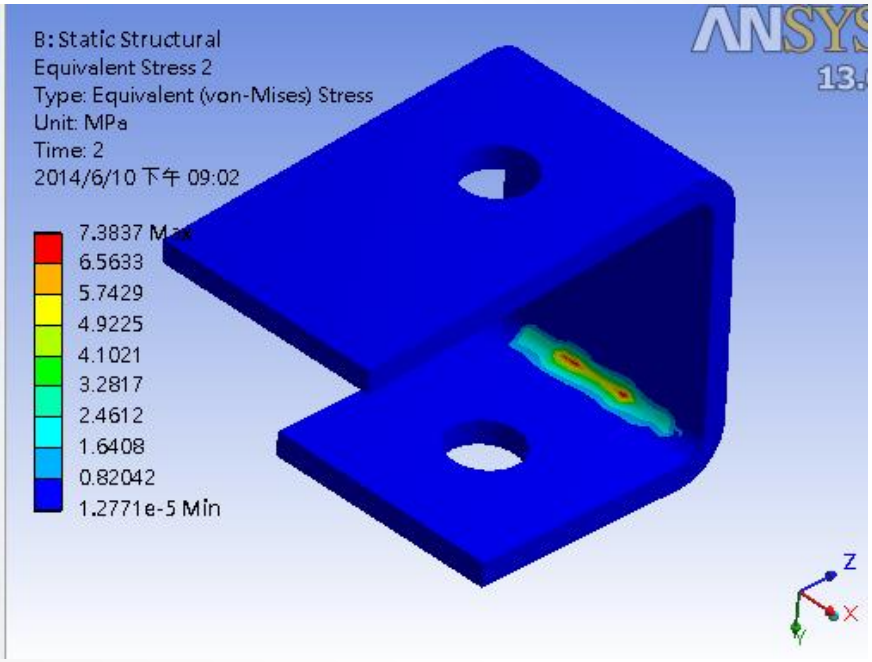
Step Controls	
Number Of Steps	2.
Current Step Number	1.
Step End Time	1. s
Auto Time Stepping	Program Controlled
Solver Controls	

Definition	
Type	Pressure
Define By	Components
Coordinate System	Global Coordinate System
X Component	Tabular Data
Y Component	Tabular Data
Z Component	Tabular Data
Suppressed	No

非線性材料模擬分析



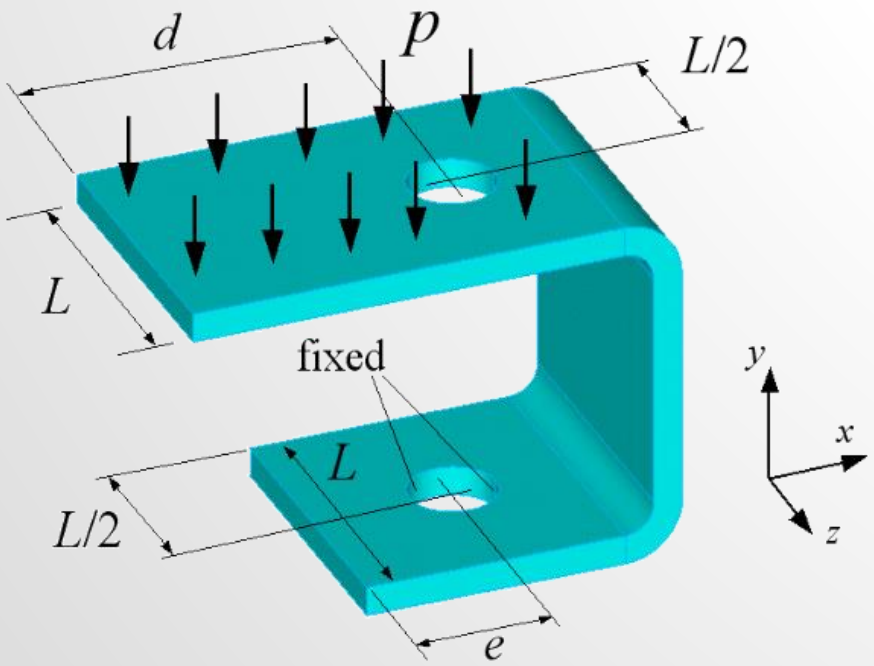
Time = 1

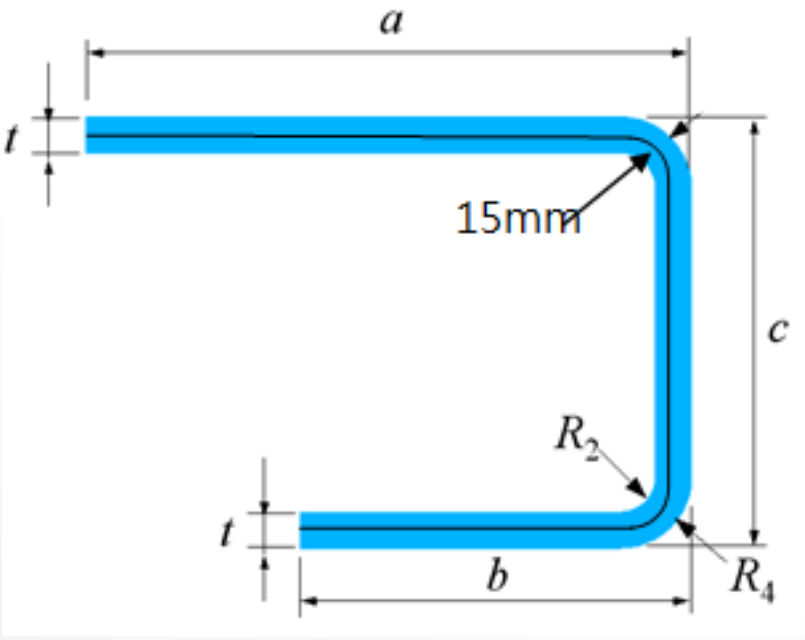


Time = 2

Exercise 16 CAE- 9-殘留應力分析
(來源：成功大學李輝煌教授)

- 如圖為一機械元件，兩圓孔半徑均為15mm，a=170mm，b=110mm，c=120mm，L=100mm，d=110mm，e=50mm，板為等厚度t=10mm。板面受壓力p=0.1MPa，底部圓孔為固定拘束。分析單位系統採用：mm、N、MPa。材料為彈塑性材料(perfectly elastic-plastic material)，楊氏係數為200GPa，浦松比為0.3，降服強度＝80MPa，slope=0 試求出塑性區域位置及殘留應力有多大。

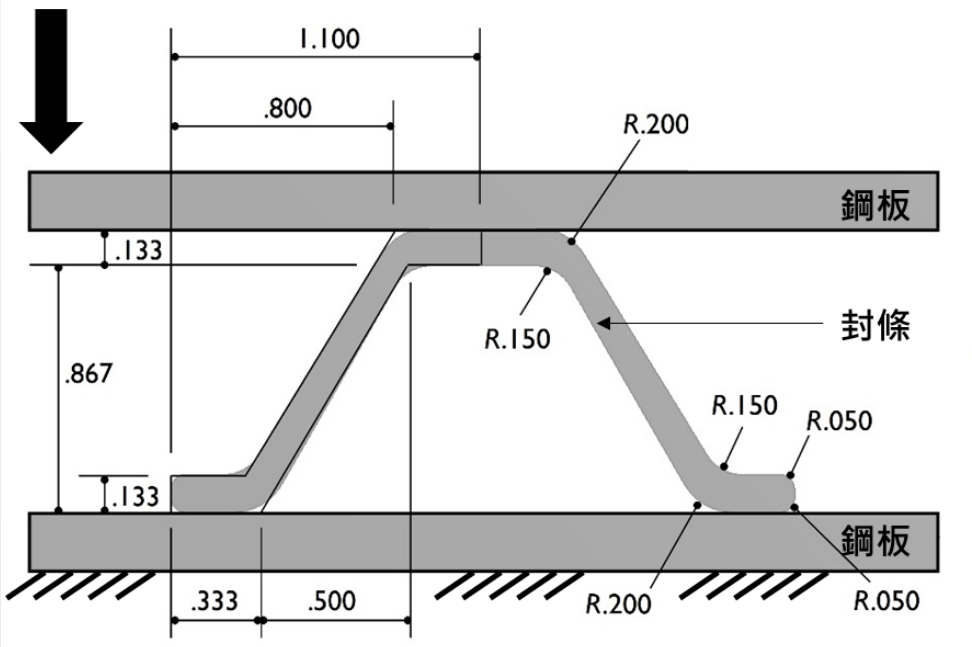




Exercise 17 CAE- 10-非線性材料
(來源：成功大學李輝煌教授)

如圖為一冰箱門封元件，由兩鋼板及一長條形封條組成。封條為超彈性材料，其材料特性由實驗量測得到(TESTDATA)，包含單軸/雙軸拉伸測試及剪力測試。本習題將學習如何藉由實驗數據輸入得到超彈性材料特性，並模擬封條受兩鋼板(E=200GPA, N=0.3)擠夾之力學行為。此次將以2D進行模型建構，並以PLANE STRAIN進行模擬後觀察其最大主應力(變)/最小主應力(變)/剪應力(變)。

位移0.85"



單位：inch